VII International Conference on Computational Methods in Marine Engineering
MARINE 2017

Nantes, France
May 15 – 17, 2017
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Preface</td>
<td>7</td>
</tr>
<tr>
<td>Acknowledgements</td>
<td>9</td>
</tr>
<tr>
<td>SUMMARY</td>
<td>11</td>
</tr>
<tr>
<td>Contents</td>
<td>13</td>
</tr>
<tr>
<td>Plenary Lecture</td>
<td>21</td>
</tr>
<tr>
<td>Invited Sessions</td>
<td>48</td>
</tr>
<tr>
<td>Contributed Sessions</td>
<td>449</td>
</tr>
<tr>
<td>Authors Index</td>
<td>1097</td>
</tr>
</tbody>
</table>
PREFACE

This volume contains full-length papers of contributions presented at MARINE 2017, the *Seventh International Conference on Computational Methods in Marine Engineering*, held at La Cité Nantes Events Center, Nantes, France, 15-17 May 2017.

The conference program includes 7 plenary lectures and 164 contributions articulated in 7 contributed sessions and 11 invited sessions organised by recognised experts. A total of 92 full-length papers have been submitted by authors and are presented in the volume.

MARINE 2017 is the seventh international conference on this topic organized in the framework of the Thematic Conferences of the European Community on Computational Methods in Applied Sciences (ECCOMAS). The first edition of this series of conferences was held in Oslo, Norway, in June 2005, with following editions every second year, in Barcelona, Spain, June 2007, in Trondheim, Norway, June 2009, in Lisbon, Portugal, September 2011, in Hamburg, Germany, May 2013, and, finally in Rome, Italy, June 2015.

In the wake of previous editions, the objective of MARINE 2017 is to provide "a meeting place for researchers developing computational methods and scientists and engineers focusing on challenging applications in Marine Engineering". The state of the art in computational approaches is addressed in sessions on computational fluid dynamics, design and optimization, fluid-structure interaction with a specific focus on ship hydrodynamics. In addition, MARINE 2017 gives special attention to themes related to marine renewable energy technologies, sailing and yacht engineering, erosion and cavitation.

MARINE 2017 is organized in the framework of the Thematic Conferences of the European Community on Computational Methods in Applied Sciences (ECCOMAS). Moreover, MARINE 2017 is a Special Interest Conference of the International Association for Computational Mechanics (IACM). The special support from NUMECA International is also gratefully acknowledged. The conference is jointly organized by Centrale Nantes, the French National Center for Scientific Research (CNRS) and by the International Center for Numerical Methods in Engineering (CIMNE) in co-operation with the Technical University of Catalonia (UPC).

Our sincere appreciation goes to plenary lecturers, invited session organizers and all authors who have contributed to the outstanding scientific quality of the conference as reflected in the proceedings. Finally, we wish to thank Mr. Alessio Bazzanella and the staff from the Congress Department of CIMNE, Barcelona, Spain, for their excellent work in the support of the conference organization and for the publication of this volume.

Nantes, 15th of May 2017

Michel Visonneau, Patrick Queutey, David Le Touzé (ECN-CNRS), Editors
ACKNOWLEDGEMENTS

The conference organizers acknowledge the support towards the organization of the MARINE 2017 Conference to the following organizations:

- Centrale Nantes (CN), France
- CNRS - National Center for Scientific Research
- International Center for Numerical Methods in Engineering (CIMNE), Spain
- European Community on Computational Methods in Applied Sciences (ECCOMAS)
- International Association for Computational Mechanics (IACM)
- NUMECA International
SUMMARY

PLENARY LECTURE ................................................................. 23

INVITED SESSIONS .............................................................. 48

IS - Deterministic and stochastic simulation-based design analysis and optimization in marine engineering .......................................................... 48

IS - Erosion and Cavitation .................................................... 143

IS - Features of Unsteady Propeller Flows ............................. 169

IS - Flow Problems and Control, Optimisation, and Uncertainty .... 220

IS - New CFD methods for marine hydrodynamics ..................... 266

IS - Numerical and experimental developments for maritime transport safety ........................................................................................................ 290

IS - Sailing and Yacht Engineering ........................................... 296

IS - Ship Hydrodynamics. Experiments and Simulations .............. 336

IS - Transient and Unsteady Effects in Ship Hydrodynamics ........ 377
GENERAL SESSIONS ............................................................................................................ 449

Design and Optimization ................................................................................................. 449

Fluid-Structure Interaction .............................................................................................. 605

Marine Renewable Energy .............................................................................................. 711

Numerical Methods in CFD ............................................................................................ 802

Offshore ............................................................................................................................ 939

Ship Hydrodynamics ....................................................................................................... 961

Structures and Composites ............................................................................................. 1087
CONTENTS

PLENARY LECTURE

High fidelity numerical simulations of ship and sub-marine hydrodynamics...... 23
C. Fureby

INVITED SESSIONS

IS - Deterministic and stochastic simulation-based design analysis and optimization in marine engineering

Are Random Coefficients Needed in Particle Swarm Optimization for Simulation-Based Ship Design? ................................................................. 48
A. Serani and M. Diez

Hull-form optimization of a 66,000 DWT bulk carrier in irregular wave condition .................................................................................................. 60
J-W. Yu, J-W. Nam, I. Lee and J-E. Choi

Hull-Form Optimization of a Luxury Yacht under Deterministic and Stochastic Operating Conditions via Global Derivative-Free Algorithms ...... 71
C. Leotardi

Mission-based hull form and propeller optimization of a transom stern destroyer for best performance in the sea environment ................. 83

Multi-objective hull-form optimization of a SWATH configuration via design-space dimensionality reduction, multi-fidelity metamodels, and swarm intelligence ............................................................................ 95
R. Pellegrini, A. Serani, S. Harries and M. Diez

Optimized DBD plasma actuator system for the suppression of flow separation over NACA0012 profile ................................................................. 107
L. Pasquale, D. Durante, M. Diez and R. Broglia

Towards the High-Fidelity Multidisciplinary Design Optimization of a 3D Composite Material Hydrofoil .................................................................................. 119
S. Volpi, M. Diez and F. Stern

Wave resistance minimisation in practical ship design ......................... 131
H. Raven and T. Scholcz

IS - Erosion and Cavitation

Improved modelling of sheet cavitation dynamics on Delft Twist 11 hydrofoil ........................................................................................................ 143
T. Lloyd, G. Vaz and A. Gnanasundaram
Numerical study of cavitation on a NACA0015 hydrofoil: solution verification
C. Negrato, T. Lloyd, T. Van Terwisga, G. Vaz and R.E. Bensow

IS - Features of Unsteady Propeller Flows

Effects of blade geometry on cavitation and pressure fluctuations of tunnel thrusters
C. Yu, X.Q. Dong, W. Li and C.J. Yang

Flow Study on a Ducted Azimuth Thruster
P. Schiller, K. Wang and M. Abdel-Maksoud

Numerical assessment of propeller-hull interaction and propeller hub effects for a twin screw vessel
H. Streckwall, Y. Xing-Kaeding, T. Lücke, T. Bugalski, T. Goedicke and A. Talay

Numerical modelling of marine cycloidal propellers for dp applications
M. Altosole, S. Donnarumma, V. Spagnolo and S. Vignolo

IS - Flow Problems and Control, Optimisation, and Uncertainty

Adjoint-based optimization methods for flows problems
T. Grahs and J. Turnow

Multi-objective surrogate based hull-form optimization using high-fidelity RANS computations
T.P. Scholcz and C.H.J. Veldhuis

Propeller nozzles design using viscous codes and optimization algorithms
S. Gaggero, D. Villa, G. Tani and M. Viviani

WEC parameters optimization by Genetic Algorithm method
A. Jabrali, R. Khatyr and J. Khalid Naciri

IS - New CFD methods for marine hydrodynamics

A new Volume-of-Fluid method in OpenFOAM
J. Roenby, B.E. Larsen, H. Bredmose and H. Jasak

Development of a new Particle Vortex Method for marine applications
E. Rossi, A. Colagrossi and D. Le Touzé

IS - Numerical and experimental developments for maritime transport safety

Centrifuge rolling test for ore liquefaction analysis
L. Thorel, Ph. Audrain, A. Neel, A. Bretschneider and M. Blanc
A New Adjustment-Free Damping Method for Free-Surface Waves in Numerical Simulations
J. Meyer, K. Graf and T. Slawig
296

Computational techniques used for velocity prediction of wing-sailed hydrofoiling catamarans
R. G. J. Flay and N. Hagemeister
312

Simulation-driven design of sailing yachts and motor boats
B. Hasubek and S. Harries
324

On the numerical convergence properties of the calculation of the flow around the KVLCC2 tanker in unstructured grids
A.L. Rocha, L. Eça and G. Vaz
336

RANS-based full-scale power predictions for a general cargo vessel, and comparison with sea-trial results
A.R. Starke, K. Drakopoulos, S.L. Toxopeus and S.R. Turnock
353

Ship Resistance Prediction - Verification and Validation Exercise on Unstructured Grids
P. Crepier
365

A generalised unsteady hybrid DES/BEM methodology applied to propeller-rudder flow simulation
D. Calcagni, F. Salvatore, G. Dubbioso and R. Muscari
377

Generation of divergence free velocity fields with prescribed integral lengths and arbitrary anisotropy for LES inlet conditions
H. Kröger, N. Kornev and P. Anschau
393

Grid generation for wall-modelled LES of ship hydrodynamics in model scale
M. Liefvendahl, M. Johansson and M. Quas
407

On the prediction of Shear-layer flows with RANS and SRS Models
G. Vaz, F.S. Pereira and L. Eça
422

Study of unsteady hydrodynamic effects in ship stern area in shallow water
I. Shevchuk and N. Kornev
440
GENERAL SESSIONS

Design and Optimization

An IMU and USBL-aided buoy for underwater localization ...................... 449
B. Allotta, F. Fanelli, J. Gelli, N. Monni, M. Pagliai, N. Palma, A. Ridolfi and M. Bianchi

Fire Performances of a Marine Bulkhead: Numerical Evaluation .............. 461
V. Drean, G. Cuffe and G. Auguin

Geometric model for automated multi-objective optimization of foils ...... 473
E. Berrini, B. Mourrain, R. Duvigneau, M. Sacher and Y. Roux

Long-term Probability Distribution of Fixed Offshore Structural Response using an Improved Version of Finite Memory Nonlinear System Procedure .. 485
N.I. Mohd Zaki, M.K. Abu Husain and G. Najafian

Multicomponent design of rotor-stator-nozzle (RSN) propulsor on AZIPODs ........................................................................................................... 496
N. Marinich, A. Yakovlev, N. Ovchinnikov and T. Veikonheimo

Operation of thrusters in arctic waters - Arctic Thruster Ecosystem .......... 508
B. Schlecht, F. Mieth, M. Kostial, T. Rosenlöcher and S. Schumann

Parametric search and optimisation of Fast Displacement Hull Forms using RANS simulations of full-scale flow ........................................... 515
E. Bretscher, S. Norris, A. Mason, G. Macfarlane and J. Denier

Propulsion Performance Optimization of “Neighbour Duct” by CFD ........ 529
K. Katayama, Y. Okada, Y. Ichinose and R. Fukasawa

Rational design of fast boat hull ............................................................. 538
N. Drimer, O. Neuberg, Y. Moshkovich and R. Hakmon

Sinkage, trim, drag of a common freely floating monohull ship ............ 550
C. Ma, Y. Zhu, H. Wu, W. Li, H. Fu and F. Noblesse

Steps towards a self calibrating, low reflection wave maker using NARX neural networks ............................................................... 562
P. Schmitt

Systematic analysis of mesh and meshless CFD Numerical Methods for water impact problems ................................................................. 568
L. Bonfiglio, S. Gaggero, A. Papetti, G. Vernengo and D. Villa

The Effect of Wave In-Deck in Conventional Pushover Analysis ............ 583
N.U. Azman, M.K. Abu Husain, N.I. Mohd Zaki, E. Mat Soom and G. Najafian

Trim optimisation in waves ................................................................. 592
A. Drouet, P. Sergent, D. Causeur and Ph. Corrignan

Fluid-Structure Interaction

Accelerated free-surface flow simulations with interactively moving bodies ............................................................................................................. 604
A.E.P. Veldman, H. Seubers, P. van der Plas and J. Helder
An innovative tool to study and optimize racing yacht appendages using fluid structure interactions ................................................................. 616
R. Balze, H. Devaux, D. Glèhen, N. Bigi, K. Roncin, J.B. Leroux and A. Nême

Application of a 6DOF algorithm for the investigation of impulse waves generated due to sub-aerial landslides .............................................................. 625
A. Kamath, H. Bihs and Ø. A. Arntsen

Development of a fluid structure coupling for composite tidal turbines and marine propellers ................................................................. 635
P. Muller and F. Pécot

Modelling of hydrodynamic loads on aquaculture net cages by a modified Morison model ................................................................. 647
P. T. Bore, J. Amdahl and D. Kristiansen

Monolithic coupling of rigid body motion and the pressure field in foam-extend ................................................................. 663
I. Gatin, V. Vukčević and H. Jasak

Numerical investigation of strong added mass effect for fluid-structure calculations applied to moving hydrofoils ........................................ 670
E. Lefrançois

Numerical prediction for many floating debris transported in city model due to Tsunami-induced flows ................................................................. 682
S. Ushijima, D. Toriu, K. Aoki, H. Itada and D. Yagyu

Simulation of breaking focused waves over a slope with a CFD based numerical wave tank ................................................................. 693
M. Alagan Chella, H. Bihs, C. Pkozdi, D. Myrhaug and Ø. A. Arntsen

Simulation of fish escape and swimming toward a predefined goal ........ 704
S. A. Ghaffari, S. Viazzo, K. Schneider and P. Bontoux

Marine Renewable Energy

A 3D RANS-VOF wave tank for oscillating water column device studies .... 710
P.R.F. Teixeira, E. Didier and G.N. Neves

An Experimental Study of the Hydrodynamic Behavior of a TLP platform for a 5MW Wind Turbine and OWC Devices .................................................. 722
G.M. Katsaounis, S. Polyzos and S.A. Mavrakos

Assessment of the parametric roll behaviour of a point absorber wave energy conversion device in irregular waves .................................................. 732
C. Meskell and K. Tarrant

Hybridization of Finite Element-Boundary Element Methods using an adaptive absorbing boundary condition for vibro-acoustic underwater noise simulations .................................................. 744
N. Zerbib, K. Bouayed, J. Lefevbre, L. Mebarek and M. Anciant

Hydrodynamic analysis of a Semisubmersible Floating Wind Turbine. Numerical validation of a second order coupled analysis ............. 765
J.E. Gutiérrez-Romero, B. Serván-Camas and J. García-Espinosa
Numerical simulation of an array of heaving floating point absorber
Wave Energy Converters using OpenFOAM .......................................... 777
B. Devolder, P. Rauwoens and P. Troch

The effects of turbulence intensity on the downstream performance
of horizontal axis tidal stream turbines .................................................... 789
I Masters, M Edmunds, A. J. Williams, P. Pyakurel and J. H. VanZwieten

A Numerical Study on the Shedding Frequency of Sheet Cavitation .......... 801
T. Melissaris, N. Bulten and T. Van Terwisga

A volume-preserving inflow boundary based numerical tank applied
to wave-structure interaction in near-shallow water ................................. 813
S. Saincher and J. Banerjee

An investigation on the effects of time integration schemes
on weakly compressible SPH method .......................................................... 825
D. C. Kolukisa, M. Ozbulut and E. Pesman

Comparative analysis of tip vortex flow using RANS and LES ...................... 836
A. Asnaghi, R.E. Bensow and U. Svennberg

Discrete element method simulation of a Split Hopper Dredger
discharging process ................................................................................. 848
J. Basic, D. Ban, N. Degiuli and N. Govender

Experimental Investigation of the Aerodynamic Flow in the
Aircraft Carrier Ski-jump by means of PIV ................................................ 861
R. Bardera, M. León-Calero and A. Rodríguez-Sevillano

Large Scale CFD Modeling of Wave Propagation into Mehamn Harbor ...... 873
W. Wang, H. Bihs, A. Kamath and Ø. A. Arntsen

Numerical study of wave transformation using the free
surface reconstruction method .................................................................... 884
A. Aggarwal, M. Alagan Chella, H. Bihs, C. Pákzodi and Ø. A. Arntsen

On the design of block preconditioners for maritime engineering ............... 893
C.M. Klaij, X. He and C. Vuik

On the use of Euler and Crank-Nicolson time-stepping
schemes for seakeeping simulations in OpenFOAM ................................. 905
S. Seng, C. Monroy and S. Malenica

Simulation of floating bodies using a combined immersed
boundary with the level set method in REEF3D ........................................ 921
H. Bihs, A. Kamath, J. Zheng Lu and Ø. A. Arntsen

Study on the Rudder Characteristics of Ultimate Rudder by
Numerical Calculation ................................................................................ 932
Y. Tendou, Y. Okada, T. Kajihama, T. Kajino, K. Katayama, M. Okazaki and K. Fukuda
Offshore

A global approximation to the Green function for diffraction-radiation of regular water waves in deep water ................................................................. 938
H. Wu, Y. Zhu, C. Ma, W. Li, H. Fu and F. Noblesse

Modelling of rotating vertical axis turbines using a multiphase finite element method .......................................................................................... 950

Ship Hydrodynamics

A numerical study of added resistance, speed loss and added power of a surface ship in regular head waves using coupled URANS and rigid-body motion ........................................................................ 960
S. Aram and S.E. Kim

Development of Numerical Method to Simulate Flows around a Ship in Regular Waves including the Effect of Ship Propulsion Plant Model ...... 975
K. Ohashi

Estimation Of Added Wave Resistance With CFD Code ........................................ 985
R. Huret, D. Causeur, A. S. Dubois, A. Drouet and O. Thilleul

Influence of the draft to ship dynamics in the virtual tank based on OpenFOAM ......................................................................................... 996
P. Du, A. Ouahsine and P. Sergent

Investigation of the confinement effect on hydrodynamic derivatives of a 135m inland containership ......................................................... 1004
I. Razgallah, S. Kaidi and H. Smaoui

Numerical assessment of interference resistance for a series 60 catamaran. ..................................................................................... 1016
A. Farkas, N. Degiuli and I. Martić

Numerical simulation of large commercial ship navigation on Paraná River, Argentina ................................................................. 1028
G. Franck, S. Mangini, P. Hugo, H. José and P. Yasser

Numerical studies on air drag reduction methods for a large container ship with fully loaded deck-containers in oblique winds ............ 1040
T. V. Nguyen, N. Shimizu, A. Kinugawa, Y. Tai and Y. Ikeda

Ship scale self propulsion CFD simulation results compared to sea trial measurements ................................................................. 1052
V. Vukčević, H. Jasak, I. Gatin and T. Uroic

System-based maneuvering simulation of a ship navigating in the confined waterway ................................................................. 1062
A. Ouahsine, P. Du and P. Sergent

Towards CFD guidelines for planing hull simulations based on the Naples systematic series ................................................................. 1071
S. Mancini, F. De Luca and A. Ramolini
Structures and Composites

A Scalable and Prismatic Pressure Vessel for Transport and Storage of Natural Gas ........................................................................................................ 1086
P. G. Bergan and D. Chang

Application of 2D Wave Spectra Time Series for Pipelaying Vessel Stinger Structural Assessment ........................................................................ 1098
A. Del Guzzo, F. Gaggiotti, C.A. Rossetti, F.S. Di Tomaso and R. Bruschi

Static analysis of a composite wind turbine blade using finite element model ........................................................................................................ 1110
M. Özyıldız and D. Çöker
HIGH FIDELITY NUMERICAL SIMULATIONS OF SHIP AND SUBMARINE HYDRODYNAMICS
MARINE 2017
CHRISTER FUREBY*

* The Swedish Defence Research Agency – FOI
Defence Security Systems Technology
SE 147 25 Tumba, Stockholm, Sweden
e-mail: fureby@foi.se

Key words: Large Eddy Simulation, hybrid RANS-LES, wall-models.

Abstract. This paper discusses the use of wall-modeled LES and hybrid RANS-LES models for the prediction of ship and submarine flows. Results from applied cases are discussed to illustrate the use of these methods for practical problems as well as the differences between methods. The paper then discusses the underlying theories and assumptions of wall-modeled LES and hybrid RANS-LES models. The focus of this presentation is on wall-modeled LES as these methods are theoretically more well-founded than hybrid RANS-LES models. Results from both canonical and building block flows are then presented and discussed in order to provide a more firm and practical foundation for the recommendations for applied use that are provided in the final concluding remarks section.

1 INTRODUCTION

Much of the current understanding of ship hydrodynamics has been obtained through systematic wind tunnel and towing tank experiments and tests in model-scale as well as trials and experiments in full-scale. Systematic and continuous analysis of experimental and test results has developed the current understanding of ship hydrodynamics, and to a significant extent also the advanced hull and propulsor designs now entering production. However, the cost of both model-scale, and in particular full-scale, tests and experiments currently precludes large systematic variation of multiple parameters in order to increase our general knowledge of ship hydrodynamics, and other methods of investigation are necessary. Only recently have Computational Fluid Dynamics (CFD) made its entrance onto the ship hydrodynamic arena, [1-2], as supported by the advancements in scientific computing, and the increasing access to large-scale massively parallel computational platforms needed for rapid turn around of simulations and very large simulations, [3]. The ability to perform a very large number of relatively simple and cheap CFD simulations (with limited details) facilitates automatic hull and propeller optimization, [4], whereas the ability to perform very large CFD simulations (with much detail) enables increased detailed understanding of the flow physics. With improved access to such tools, new experimental challenges are posed, focusing on revealing and quantifying details of the flow in order to validate the CFD models, e.g. [5].

The flow around a ship or submarine hull is dominated by the boundary layer that develops over the hull, but is also influenced by the pressure gradients developing over the bow and the
C. Fureby

stern as well as the disturbances caused by appendages, struts, bilge-keels, rudders and the propeller(s). CFD simulations require the volume of interest to be subdivided into control volumes, the size of which represents the flow structures to be resolved by the simulation. For the sake of discussion we consider two ships, a small ship with a length of 50 m, a width of 12 m and draught of 4 m, moving at 10 knots, and a large ship with a length of 300 m, a width of 30 m and draught of 10 m, moving at 30 knots. The global Reynolds numbers (measuring the ratio of inertial forces to viscous forces, indicating the level of turbulence) of these ships are Re0 = v0L/ν = 2.5·10^8 and 4.6·10^9, respectively, in which v0, L and ν are the speed, hull length and viscosity. At model-scale, the Re0 numbers are approximately two orders of magnitudes smaller, whereas the integral Re number, ReI, (representative of the most energetic eddy turbulence scales) can be assumed one order of magnitude smaller than Re0. Based on the present understanding of turbulence, [6-9], the ratio of the integral scales, ℓI, to the Kolmogorov scales, ℓK, scale with ReI such that ℓI/ℓK = ReI^{3/4}, which means that the Kolmogorov scales are ℓK ≈ 0.13 mm and 0.09 mm, respectively. In Direct Numerical Simulation (DNS) (in which all turbulent scales are resolved), [10], the grid resolution, ∆, should be smaller than ℓK, which implies that ∼2.1·10^11 and ∼6.6·10^{12} surface grid cells are needed to resolve the surface flow. If we assume that the computational domain extends one hull-length away from the hull in all directions, and if geometrical progression is used to expand the cells from the hull to a distance of 10% of the hull length, thereafter uniform cells are used, ∼56500 and ∼348000 cells are required in the wall-normal direction. All in all, this results in that ∼5.0·10^{11} and ∼1.0·10^{13} grid cells are required for DNS of the corresponding model-scale hulls. These estimates suggest that DNS is unachievable for both model-scale and full-scale ship and submarine hydrodynamics for some time.

The traditional alternative to DNS is to employ Reynolds Averaged Navier-Stokes (RANS) models, [11-12], which is based on computing the mean (time-averaged or ensemble averaged) flow using models, [12-14], for the whole spectrum of the turbulence. RANS has been successfully used for decades to compute the mean flow around many hull forms, and is also widely used in aerodynamics and other related fields. The main drawbacks of RANS are that it cannot handle large-scale unsteadiness, and that the method is not designed to provide information about unsteady features of the flows or small details of the flows. The grid resolution is primarily dictated by the turbulence model selected, [14], but is usually on the order of ∼1·10^6 to ∼30·10^6 grid cells for a full-scale ship hull.

An alternative to DNS and RANS is Large Eddy Simulation (LES), [15-20], in which only the large energetic eddy scales are explicitly simulated, using a subgrid model to represent the effects of the small-scale turbulence. If we assume that the Taylor scales, [7-9], should be resolved in LES, an estimate, based on the perception that the Taylor scales gradually reduce to the Kolmogorov scales in the wall-normal direction as the hull is approached (resulting in anisotropic near-wall scales) results in that ∼5.0·10^{11} and ∼1.0·10^{13} grid cells are needed for a wall-resolved LES of the full-scale hulls, whereas ∼1.2·10^8 and ∼1.6·10^9 grid cells are required for wall-resolved LES of the model-scale hulls. Wall-resolved LES of full-scale ship flows are thus unfeasible but wall-resolved LES of model-scale ship flows are within reach as demonstrated by Posa & Balares, [21]. Analysis, [22-24], suggests that more than 90% of these grid cells are employed to resolve the gradually decreasing eddy scales in the boundary layer. If it can be assumed that the inner part of the boundary layer is ‘enslaved’ by the outer part of the boundary layer, and hence to behave in a deterministic manner relative to the outer
part of the boundary layer, a model may be used to replace the explicit simulation of the inner part of the boundary layer. Such a simulation approach will hereafter be referred to as a *wall-modeled* LES in contrast to the wall-resolved LES in which the flow in the whole boundary layer is explicitly simulated. Wall-modeled LES thus has the potential of being able to tackle full-scale ship hydrodynamics with available computational resources without significant decrease in accuracy compared to wall-resolved LES, [25]. A similar approach is hybrid RANS-LES, [26-27], which attempts, using different schemes, to merge RANS in the (inner) attached part of the flow with LES in the (outer) detached part of the flow.

In this review we will discuss and compare DNS, wall-resolved LES, wall-modeled LES, hybrid RANS-LES and RANS in order to increase the understanding and usefulness of these complementary methods. Both applied and canonical flows will be discussed using example references idiosyncratic to the author and his colleagues.

### 2 TARGET CONFIGURATIONS

Here we discuss major flow features of typical target configurations such as flows around surface ships and submarines at straight-ahead and yaw conditions. These discussions are here based on RANS, hybrid RANS-LES and wall-modeled LES in order to illustrate the spectrum of flow physics features that can be predicted by these methods.

Figure 1 shows the flow around a 1:58 scale model of the KRISO Very Large Crude Carrier (KVLCC2), which in full scale has a length of 320.0 m, a width of 58.0 m, a draught of 20.8 m and a displacement volume of 312622 m$^3$, originally introduced by Van et al., [28]. Comprehensive experimental data are available, ranging from towing tank measurements, [28-31], to wind tunnel measurements, [32-33]. There are also a significant number of computational studies, [34-37], performed on this hull form using different CFD models. More specifically Petterson *et al.*, [37], used this hull form to evaluate RANS, hybrid RANS-LES and wall-modeled LES, and to further elucidate the flow physics at straight ahead conditions and 12° and 30° of drift at global Re numbers of Re$_0$=3.70·10$^6$ and 4.60·10$^6$. The RANS computations were performed with ReFRESCO, [38], using the 2003 version of Menter’s SST k-$\omega$ model, [39], and block-structured O-O grids with 12.7 Mcells in a large cylindrical computational domain with a normalized wall distance of $y^+<1$. Here, $y^+ = u^+\nu/u_τ$ is the normalized distance to the wall in which $u_τ = \sqrt{\tau_w}$ is the friction velocity and $\tau_w$ the wall shear stress. The hybrid RANS-LES were performed with EDGE, [40], using an algebraic turbulence model based on combining a mixing length RANS model, [16], with the Smagorinsky model, [42]. Unstructured grids with 73.5 Mcells and $y^+<1$ were used. The wall modeled LES were performed with OpenFOAM, [43], using the mixed model, [44], a wall model, and unstructured grids with 121, 194 and 202 Mcells depending on the drift angle, with $y^+\approx10$.

The flow images in figure 1 show the vorticity distributions around the KVLCC2 hull from RANS, hybrid RANS-LES and wall-modeled LES at 0°, 12°, and 30° drift, respectively, in terms of iso-surfaces of the second invariant of the velocity gradient tensor, $λ_2$. At 0° drift the flow is dominated by the boundary layer over the forebody, parallel mid-section and stern. LES reveals a pair of flat bilge vortices, developing at the bow, extending towards the sides of the hull, whereafter they become unstable and participate in forming the aft-body vortices, not observed in RANS or hybrid RANS-LES. Similar vortex shapes are however observed at the midship cross-section in the experimental results of Lee *et al.*, [32]. The flow over the stern is very complex due to the combined influences of geometry, pressure gradient and viscosity,
Compared to the RANS, which only predicts the mean aft-body vortices, the hybrid RANS-LES and the LES predicts a complex vorticity field evolving around the unsteady undulating aft-body vortices containing also Ω-shaped vortices developing sideways to the hull. With gradually increasing drift angle, the flow rapidly becomes more complicated: At 12° drift new vortical systems such as a forebody bilge vortex, a forebody side vortex, a stern vortex, an aft-body side vortex, an aft-body bilge vortex and aft-body hairpin vortices develops, whereas the aft-body vortices have been combined into the aft-body side vortex on the windward side and the aft-body hairpin vortex on the leeward side. The unsteadiness also becomes more evident as can be observed from the hybrid RANS-LES and the LES predictions. At 30° drift the fore-body bilge vortex, forebody side vortex, aft-body side vortex, aft-body hairpin vortex and the aft-body bilge vortex are very well established. A stern vortex can also be found between the aft-body side vortex on the windward side and the aft-body hairpin vortex on the leeward side. As observed also in the LES of the 12° drift case, oblique bilge vortices are observed aft of the fore-body bilge vortex. From the LES predictions these vertical structures are surrounded by secondary vortical structures resulting from Kelvin-Helmholtz vortices and von-Karman shedding over the fore-body and helical instabilities.

The time-averaged axial velocity profiles in figure 1 compares RANS, hybrid RANS-LES and LES predictions in the propeller plane with experimental data from [32] at 0° drift. The most pronounced feature in the axial velocity distribution is the hook-shaped pattern of low-velocity around the propeller boss and the low-velocity regions on either side of the transom part of the hull. These features are captured by all simulation models, but most accurately by the wall-modeled LES. The RANS and hybrid RANS-LES predictions either overpredict or underpredict the size and strength of these features. The hook-shaped flow pattern results from the counter-rotating after-body vortex pair developing on each side of the tapered part of

![Figure 1: Flow around the KVLCC2 hull in model scale seen from below at 0° drift (top), 12° drift (middle) and 30° drift (bottom). RANS predictions are presented in the left column, hybrid RANS-LES in the middle column, and wall modeled LES in the right column, [37]. In all cases the flow is illustrated using iso-surfaces of the second invariant of the velocity gradient tensor, \( \lambda_2 \). In the three lowest panels the time-averaged axial velocity at 0° drift in the nominal propeller plane is compared with the experimental data from [32].](image-url)
the lower hull gradually developing over the propeller boss, which in turn is influenced by the low-velocity regions on either side of the stern transom. The low-velocity regions on either side of the stern result from the change in cross-sectional hull area and the large adverse pressure gradient, and gradually thickening boundary layer. This results in the development of the multiple hairpin and Ω-shaped vortex structures found in the LES predictions.

Figure 2 shows the flow around the DSTG Joubert generic submarine model created from the work of Joubert, [45-46], who proposed a design based on principles consistent to achieve low resistance and flow noise, in particular over the forward sonar. The hull-form comprises an axisymmetric body of revolution with a length to diameter ratio of 7.3 and a length of 70.2 m. The bow shape provides a small negative pressure coefficient and pressure gradient, while keeping the location of pressure minima as far aft as possible. Several experimental and computational studies of this hull-form have been conducted, [47-50]. In [47] wind-tunnel experiments were combined with RANS and wall-modeled LES to investigate the flow around the boat in model scale at straight-ahead conditions, whereas in [48] this research was extended to also include effects of yaw. Unstructured grids with between 209 and 340 Mcells were used, with surface averaged $y^+$ values of ~20. Good agreement between experimental data, comprising surface flow characterization using tufts, pressure distribution along the meridian plane and Particle Imaging Velocimetry (PIV) data in ‘stitched’ patches at the meridian plane and in

![Figure 2: Flow around the fully appended Joubert submarine model in full scale at straight ahead conditions at 9.2 knots with a 5-bladed propeller (top) and a 7-bladed propeller (bottom), [50]. The flow is illustrated with isosurfaces of the second invariant of the velocity gradient tensor, $\lambda_2$. At the two lower panels the time-averaged axial velocity, $\langle v_x \rangle / v_0$, and its rms fluctuations, $v_{x,\text{rms}} / v_0$, are presented in various cross-sections along the hull for the two propeller configurations and for the two boat speeds of 4.6 and 9.2 knots.](image-url)
several cross-sectional planes, [47-49], and RANS as well as wall-modeled LES were demonstrated. The combined experimental and computational flow predictions provided a comprehensive picture of the flow physics that was summarized in [48]. In [49] this research was further extended to include also propulsion effects. More specifically, numerical simulations using wall-modeled LES were performed for a full-scale version of the DSTG Joubert generic submarine equipped with a 5-bladed propeller (DSTG 115-1) and a 7-bladed propeller (DSTG 057-1), and simulations were performed at two speeds, 4.6 and 9.2 knots. For the wall modeled LES, unstructured grids were employed with between 209 and 309 Mcells, having surface averaged $y^+$ values of $\sim 100$ and $\sim 200$ depending on the speed. Particular refinement patches were added in the stern region, around the propeller and in the slipstream.

The top and middle panels of figure 2 show the flow around the full-scale DSTG Joubert generic submarine in terms of iso-surfaces of the second invariant of the velocity gradient tensor, $\lambda_2$, colored by the axial velocity. Near the intersection of the fin and the casing, the flow rolls up into a horseshoe-vortex system that surrounds the base of the fin and extends downstream along the casing until it interacts with other vortices in the stern, and with the wake of the fin. Standing side-vortices are formed towards the trailing edge of the fin that interact with the unsteady hull boundary layer and the horseshoe vortex, creating an unsteady wake behind the fin. The flow over the fin-cap is dominated by two sets of vortical structures – a main pair of counter-rotating vortices that develop at the widest section of the fin-cap, separate from the trailing edge of the fin-cap and persist far downstream to the stern region; and a secondary pair consisting of a vortex on each side of the fin just below the cap. This secondary pair is seen to separate from the trailing edge of the fin, below the fin cap, and continue downstream with a small deviation towards the casing. In contrast, the primary vortex pair appears to deviate away from the casing as they travel downstream. Towards the stern, the legs of the horseshoe vortex system pass between the upper rudders whilst interacting with the innermost leg of the horseshoe- vortex system developing around the upper rudders. Additional horseshoe-vortex systems develop around the lower rudders. Tip vortices are shed from the propeller blades and are seen to persist for a number of propeller diameters until they gradually break-up. A central vortex extending downstream of the hub is also produced by the propeller and maintains its form much further aft than the blade-tip vortices.

The two bottom panels of figure 2 compare the predictions of the time-averaged axial velocity, $\langle v_x \rangle /v_0$, and the axial rms velocity fluctuations, $v_{x,\text{rms}}/v_0$, at some axial locations from $x/L=0.550$ to $x/L=1.105$. Included in the comparisons are all four wall-modeled LES predictions with both propeller configurations at both boat speeds. Regarding $\langle v_x \rangle /v_0$, it is seen that the velocity is virtually unaffected by the choice of propeller and only marginally influenced by the speed. The boundary-layer development and the gradually weakening imprints of the fin-tip vortex pair and secondary fin-tip vortex system are not affected by either the propeller or the speed. Even the time-averaged flow over the stern appears almost unaffected by the choice of propeller. The $\langle v_x \rangle /v_0$ distribution across the slipstream, however, reveals that that 5-bladed DSTG115-1 propeller results in a more intense hub-vortex. Regarding $v_{x,\text{rms}}/v_0$, elevated fluctuation levels are observed in the wake behind the fin and in the hull boundary layer. Along the tapered stern part of the hull, the region of high velocity fluctuations gradually widens as the horseshoe vortex structures interact with the boundary layer. Only small differences in $v_{x,\text{rms}}/v_0$ can be observed along the stern, but $v_{x,\text{rms}}/v_0$ increases with increasing speed. Moreover, the stronger tip vortex of the 5-bladed DSTG115-1 propeller also results in higher levels of $v_{x,\text{rms}}/v_0$ that stretches far downstream in the slipstream.
3 MATHEMATICAL MODELING

Next, we will outline the frameworks involved in the mathematical modeling of turbulent flows in general and near-wall flows in particular. This modeling survey will be brief, and for details of different methods and implementations we refer to the original papers.

3.1 Governing Equations, DNS, LES, Hybrid RANS-LES and RANS

Irrespective of modeling approach, DNS, wall-resolved LES, wall-modeled LES, hybrid RANS-LES or RANS, the governing equations are the Navier-Stokes equations,

\[ \partial_t (v) + \nabla \cdot (v \otimes v) = -\nabla p + \nabla \cdot S, \quad \nabla \cdot v = 0, \]  

in which \( v \) is the velocity, \( S = 2\nu D \) the rate of strain tensor, \( \nu \) the viscosity, and \( D = \frac{1}{2}(\nabla v + \nabla v^T) \) the rate of strain tensor, in which \( \nabla v \) is the velocity gradient tensor.

In RANS the Navier-Stokes equations (1) are averaged over an ensemble of equivalent flows, \( \langle v \rangle (x) = \frac{1}{N} \sum_{i=1}^{N} v_N(x,t) \), or equivalently over time, \( \langle v \rangle (x,t) = \frac{1}{T} \int_0^T v(x,t) dt \), so that,

\[ \partial_t \langle v \rangle + \nabla \cdot (\langle v \rangle \otimes \langle v \rangle) = -\nabla \langle p \rangle + \nabla \cdot (\langle S \rangle - R), \quad \nabla \cdot \langle v \rangle = 0, \]  

in which \( R = \langle v' \otimes v' \rangle \) is the Reynolds stress tensor, representing the transport of momentum due to the velocity fluctuations \( v' \). In order to close (2) and to represent the effects of the turbulence, \( R \) must be modeled. A common method of approximating the \( R \) is based on the hypothesis that the effects of turbulence are analogous to an increased viscosity. This is justifiable when effects such as energy dissipation and increased mass transport normal to mean flow streamlines are considered. The Boussinesq relationship, [51], between the Reynolds stresses and the mean flow strain embodies this approximation and is formulated as

\[ R = -2\nu_t \langle D \rangle, \]  

in which \( \nu_t = \frac{c\mu k^2}{\epsilon} \), and \( \langle D \rangle \) the mean rate-of-strain tensor. Many different closure models for \( R \) are available, [12, 52], including algebraic, one-equation, two-equation and differential stress equation models. The most widely used RANS turbulence models are the \( k-\epsilon \) model, [53, 12], in which \( \nu_t = \frac{c_a k^2}{\epsilon} \), and the shear stress transport (SST) \( k-o \) model, [39], in which \( \nu_t = \frac{a_1 k}{\max(a_1 \omega, F_2 \langle |\nabla D||\rangle)} \). Here, \( k \) is the turbulent kinetic energy, \( \epsilon \) the dissipation rate, and \( \omega \) the specific dissipation rate, all of which are obtained from modeled transport equations. For further details we refer to [12, 39, 52-53]. Most of these models are available in two versions: one for use with grids that have a wall-normal resolution of \( y^+ \leq 1 \), and one for use with grids that have a wall-normal resolution of \( y^+ > 30 \), and then in conjunction with a RANS wall-model, [52], that relates the wall shear stress, \( \tau_w \), to the mean or time-averaged velocity \( \langle v \rangle \) adjacent to the wall, using additional physical relationships.

In wall-resolved and wall-modeled LES, the Navier-Stokes equations (1) are low-pass filtered, [15], using a convolution operator of the form \( v(x,t) \rightarrow \int G_\Delta(x-x') v(x',t) dx' \) to remove the small subgrid scales (assumed more universal), [15-19], so that,

\[ \partial_t \langle \vec{v} \rangle + \nabla \cdot (\langle \vec{v} \rangle \otimes \langle \vec{v} \rangle) = -\nabla \langle \vec{p} \rangle + \nabla \cdot (\langle \vec{S} \rangle - \vec{B}), \quad \nabla \cdot \langle \vec{v} \rangle = 0, \]  

in which \( \vec{B} = \langle \vec{v} \otimes \vec{v} - \vec{v} \otimes \vec{v} \rangle \) is the subgrid stress tensor, representing unresolved transport of momentum on the resolved flow, [15-19]. In order to close (3) and to represent the physics of the unresolved flow, \( \vec{B} \) must be modeled. According to Sagaut, [15], models for \( \vec{B} \) can generally
be divided into functional and structural models depending on if they are intended to mimic the kinetic energy cascade from large to small eddy scales, usually assumed to be of inertial sub-range character, or if they are intended to mimic the structure of the subgrid flow physics.

A wide variety of LES subgrid models are available, and Sagaut, [15], provides a comprehensive summary and review of these two model classes. Functional models are more frequently used than structural models, and are also generally more robust, since most functional models are based on a Boussinesq relationship, so that $B=2\nu_k\tilde{D}$, in which $\nu_k$ is the subgrid viscosity.

A wide range of subgrid viscosity models are available including algebraic models such as the well-known Smagorinsky (SMG) model, the Wall-Adapting Local Eddy (WALE), and the $\sigma$-model, as well as one-equation models such as the One Equation Eddy Viscosity (OEEVM) model and the Localized Dynamic k-equation Model (LDKM). For further details we refer to [15] and references therein. Note that whereas wall-resolved LES only need the subgrid models, wall-modeled LES typically need additional models to represent the near-wall flow physics that is not explicitly resolved on the grid. The modeling of the near-wall flow physics in wall-modeled LES is significantly more challenging than in RANS since in LES we are resolving a significant part of the unsteady eddy motion in the boundary layer, and particularly in the outer part of the boundary layer where the flow is more energetic.

RANS have been successful in simulating wall bounded flows for decades, providing, however, only the mean velocity, $\langle \tilde{v} \rangle$, and the modeled turbulent stresses, $\mathbf{R}$, whereas LES, providing instantaneous flow realizations of the resolved velocity, $\tilde{v}$, as well as any statistical moment of $\tilde{v}$, have been considered too expensive for practical applications unless used with a wall model. Wall-modeled LES have been less-well understood and is questioned due to the additional complexity of modeling only a fraction of the near wall flow physics, whereas the remaining fraction of the near wall flow physics is resolved on the grid. As an alternative, the class of hybrid RANS-LES has evolved in which the attached flow is treated by RANS and the detached flow is treated by LES, [14, 26-27], and references therein. Hybrid RANS-LES suffers from the well-known issues of how to unambiguously define the attached and detached flow regions and how to create grids that support both RANS and LES in the two regions, as well as the transition between these two regions, [26-27]. Improved understanding of the flow physics as well as the governing equations have resulted in improved hybrid RANS-LES models that better handle different grid topologies and inappropriate grid resolution. An issue not discussed is that of how the governing hybrid RANS-LES equations are formulated: That both the RANS and LES equations (2) and (3), respectively, have the same mathematical appearance is thus taken as a basis for the hybrid RANS-LES equations,

$$
\partial_t(\tilde{v})+\nabla \cdot (\tilde{v} \otimes \tilde{v})=-\nabla p+\nabla \cdot (\tilde{S}-\mathbf{T}), \quad \nabla \cdot \tilde{v}=0,
$$

in which $\mathbf{T}=(\tilde{v} \otimes \tilde{v} - \tilde{v} \otimes \tilde{v})$ is the hybrid RANS-LES stress tensor. The bridge between the LES and RANS formulations can be expressed by the definition of the hybrid RANS-LES variables $\tilde{v}(x,t)=(G_3 \ast G_T)\tilde{v}(x,t)=\int \int_D G(x-x',\Delta)\tilde{v}(x',t)d^3x'dt$, in which $G_3 \ast G_T$ is a spatio-temporal filter kernel that may be able to distinguish between attached (or RANS) flow regions, in which $\tilde{v}(x,t)=G_3 \ast \tilde{v}(\tilde{v})(x,t)$, and detached (or LES) flow regions, in which $\tilde{v}(x,t)=G_3 \ast \tilde{v}=\tilde{v}(x,t)$. Two conceptually different hybrid RANS-LES model types are commonly referred to: The first approach can be divided into two branches in which the first branch is based on Speziale’s, [54], ideas that $\mathbf{T}=f_3(\Delta/\ell_k)\mathbf{R}$, in which any RANS turbulence model, $\mathbf{R}$, can be used, and $f_3(\Delta/\ell_k)$ is the contribution function, the role of which is to damp the contribution of $\mathbf{R}$, since part of...
the turbulence is resolved in the regime where the solution becomes unsteady. One suggested contribution function, [53], is 
\[ f_{\Delta} = (1 - \exp(-\beta \Delta / \ell_K))^n, \]
where \( \beta = 0.001 \) and \( n = 1 \). The second branch is based on a weighted sum of LES and RANS models so that for eddy-viscosity and subgrid viscosity models, the hybrid RANS-LES stress tensor is 
\[ T = -2 \nu_{hRL} \mathbf{D}, \]
in which the hybrid RANS-LES viscosity is 
\[ \nu_{hRL} = \psi \nu_t + (1 - \psi) \nu_k, \]
in which \( \nu_t \) is the RANS eddy-viscosity, \( \nu_k \) the LES subgrid viscosity, and \( \psi \) the blending factor. In this approach any RANS turbulence and LES subgrid viscosity model can be used. Typically, the blending factor depends on different solution dependent parameters from the RANS and LES models including the filter width, \( \Delta \), and the distance to the wall, \( d \). The second approach in commonly referred to as Detached Eddy Simulation (DES), [14, 26-27], and is based on a unified approach in which the same model is used both for the attached (RANS) and detached (LES) flow regions. The discriminating factor is that the distance, \( d \), to the wall is typically replaced by a simple switch function of the form 
\[ d = \min\{d, c_{DES} \Delta\}. \]
Close to the wall, where \( d < c_{DES} \Delta \), the model utilize the original RANS model. Away from the wall, were \( d > c_{DES} \Delta \), the model turns into a subgrid model. The original formulation is based on the Spalart-Allmaras model, [55], that turns into the well-known Smagorinsky subgrid model, [15], in the limit of LES.

3.2 Near-Wall Flow Physics

The general understanding of a turbulent boundary layer has been known since the 1950’s and it is composed of virtually chaotic fluid motion that results in pressure fluctuations at the wall surface. Figure 3 is a schematic of a turbulent boundary layer that shows the irregular division between the turbulent and freestream flow and the flattened shape of the mean velocity as a function of the distance from the wall, [56]. It is generally assumed that the pressure and velocity fluctuations disappear outside of the intermittent edge of the boundary layer. Experimental data and DNS predictions agree on that the boundary layer can be divided into an inner layer, which in turn is composed of a viscous sub-layer, dominated by viscous stresses, a buffer layer, in which both viscous and turbulent stresses are important, and a log-law region, which is dominated by turbulent stresses, and an outer layer, which is dominated by the external flow and the large-scale turbulence, [57]. As evident from the visualizations in [57], the turbulence kinetic energy is primarily carried by eddies of different characteristic sizes in the different layers near and far from the wall. This reveals that the turbulent boundary layer is a multi-scale phenomenon that any form of modeling needs to respect.

![Figure 3: Flow physics in a turbulent boundary layer](image-url)
The key aspects of turbulent boundary layers are most easily explained using the ensemble averaged incompressible Navier-Stokes equations (2). For the viscous sublayer, $y^+<5$, equation (2) simplifies to $\nu \frac{\partial^2 \langle v_x \rangle}{\partial y^2} = \frac{\partial \langle p \rangle}{\partial x}$. By integrating this equation twice with respect to $y$, assuming that $\frac{\partial \langle p \rangle}{\partial x}$ is independent of $y$, and after introducing the wall-shear stress component, $\tau_w = \nu \left( \frac{\partial \langle v_x \rangle}{\partial y} \right)_{y^+0}$, and the normalized streamwise velocity, $v^+_x = \langle v_x \rangle / u_*$, and wall distance, $y^+ = u_*/\nu$, with $u_* = \tau_w^{1/2}$ being the friction velocity, we obtain that,

$$v^+_x = y^+ + \frac{\nu}{2u_*^2} \frac{\partial \langle p \rangle}{\partial x} (y^+)^2.$$  \hspace{1cm} (5)

For the log-law region, $60 < y^+ < 300$, equation (1) becomes $\partial R_{xy}/\partial y = -\frac{\partial \langle p \rangle}{\partial x}$, By integrating this equation with respect to $y$, assuming that $\frac{\partial \langle p \rangle}{\partial x}$ is independent of $y$, and moreover that $R_{xy} = -\nu \left( \frac{\partial \langle v_x \rangle}{\partial y} \right)$, in which the turbulent viscosity is modeled as $\nu_t = \kappa u_*$, with $\kappa$ being the von-Karman constant, we obtain, after rearrangement and the subsequent introduction of the non-normalized velocity $v_x = \langle v_x \rangle / u_*$ and wall distance $y^* = u_*/\nu$, that,

$$\frac{\partial v^+_x}{\partial y^*} = \frac{1}{\kappa y^*} + \frac{\nu}{\kappa u_*^2} \frac{\partial \langle p \rangle}{\partial x}.$$  \hspace{1cm} (6)

By integrating equation (6) we finally obtain that,

$$v^+_x = \frac{1}{\kappa} \ln(y^*) + \frac{\nu}{\kappa u_*^2} \frac{\partial \langle p \rangle}{\partial x} y^* + B.$$  \hspace{1cm} (7)

For a zero pressure-gradient boundary layer the von-Karman constant is $\kappa \approx 0.41$, whereas $B \approx 5.2$. In the intermediate buffer-layer, $5 < y^+ < 60$, the flow physics dominating the viscous sublayer gradually transitions into the flow physics dominating the inertial sub-layer, and hence no analytical model exists for this layer. Figure 3 shows a typical turbulent boundary layer velocity profile in which the viscous and log-law regions are presented as dashed lines together with the well-known Spalding’s law-of-the-wall curve fit, [59],

$$y^* = v^+_x + e^{-(\kappa B)} \left[ e^{(\kappa y^*)} - 1 - (\kappa y^*) - \frac{1}{2!} (\kappa y^*)^2 - \frac{1}{3!} (\kappa y^*)^3 + \ldots \right],$$  \hspace{1cm} (8)

designed using experimental data to provide a continuous velocity profile through a zero pressure gradient boundary layer. The continuous velocity profile (8) asymptotically agrees with (5), when $\frac{\partial \langle p \rangle}{\partial x} = 0$, in the viscous sublayer and (7) in the log-law region, and thus provides a foundation for developing a deterministic model of the near wall flow.

### 3.3 Wall-Modeled LES

The philosophy of wall-modeled LES is to respect the multi-scale nature of the boundary layer, by directly resolving the energetic eddies in the outer layer (where they are large) but to model the energetic eddies in the inner layer (where they are small). As a consequence of this, the inner-layer dynamics (streaks, quasi-streamwise vortices, peak production and dissipation, etc.) is completely or partly removed from the dynamical system explicitly computed and represented by a single value of the wall shear stress, $\tau_w$, that the models is supposed to provide.
This is a severe truncation of the wall-turbulence dynamical system – as it has to be in order to drastically reduce the computational cost – and since the model is also required to handle a situation where parts of the inner-layer dynamics is resolved, the model needs to be very flexible and be able to take a passive role compared to the evolution of the resolved flow structures. Given the removal of the peak production region in the inner layer (typically occurring at \( y^+ \approx 12 \)) turbulence can no longer be produced in the modeled boundary layer, and thus all resolved outer-layer turbulence needs to be produced (and later dissipated) in the outer layer, as expected in high-Re wall-turbulence, where the Reynolds stresses have been found to be predominantly produced at the same wall distance as they are later dissipated, [60].

At a first glance this understanding of wall-modeled LES also incorporates hybrid RANS-LES models. There is, however, an important and discriminating distinction between wall-modeled LES and hybrid RANS-LES models in that for wall-modeled LES the LES equations (3) apply all the way down to the wall, whereas for hybrid RANS-LES models the LES equations (3) apply only above a certain ‘interface’ \( y_{int} \) (which may be defined implicitly, but nevertheless exists) below which the RANS equations (2) apply. The distinction between hybrid RANS-LES and wall-modeled LES is subtle but important: In wall-modeled LES, a wall-model is used to estimate \( \tau_w \) but the coupling between the LES and wall-model is rather weak: the LES feeds velocity data to the wall-model at \( y=h_{wm} \), and the wall-model feeds wall-shear-stress data back to the LES at \( y=0 \). Apart from this, no other information is exchanged. Notably, while the LES could impart flow structures onto the wall-model, the ability of flow structures in the wall-model to enter the LES region is limited. Perhaps more importantly, the formal definition of the LES equations applying all the way down to the wall implies that the LES equations and the wall-model overlap for a distance of \( h_{wm} \).

Two different types of wall modeled LES are currently employed: one-equation and algebraic. In one-equation wall-models, [61-62], the thin boundary layer equations, [12],

\[
\frac{\partial}{\partial y}[(v+\nu_s)\frac{\partial v_s}{\partial y}]=0 \quad \text{with} \quad \nu_s=\nu exp(y^+/A^+),
\]

in which \( A^+ \approx 17 \), are solved on an auxiliary one-dimensional wall-normal grid extending away from the wall into the LES domain. Equation (9) is solved over the region \( 0 \leq y \leq h_{wm} \), using a no-slip condition at the wall, \( y=0 \), and with \( v_s \) equal to the LES wall-parallel velocity, \( \overline{v}_p \), at \( y=h_{wm} \). The wall-shear stress \( \tau_w \) from the wall-model (9) is computed from \( \tau_w=\nu(\frac{\partial v_s}{\partial y})_w \). The next step is to construct the full wall-shear stress vector \( \tau_w \) by assuming that it is aligned with the velocity parallel to the wall, and by using a simple linear approximation for the wall-normal velocity (for the wall-normal stress). The final step is to couple this back to the LES equations (3) which is often done by using \( \tau_w \) as a boundary condition for the LES momentum equation. In algebraic wall-models, [63], an even simpler procedure is followed, using Spalding’s law-of-the-wall. Given the LES velocity in the grid cells adjacent to the wall, \( \overline{v}_p \), the friction velocity, \( u_t \), and hence also the wall-shear stress, \( \tau_w \), are computed by solving (8), in which \( v_s \) is substituted for \( \overline{v}_p/u_t \), for each grid cells adjacent to the wall,

\[
y^+ = \frac{\nu_p}{u_t} + e^{-k_\nu y^+} - \frac{1}{2}(k \nu/v^+)^2 - \frac{1}{8}(k \nu/v^+)^3 + \ldots
\]

The next step is to construct the full wall-shear stress vector \( \tau_w \) by assuming that it is aligned with the velocity parallel to the wall, and by using a simple linear approximation for the wall-
normal velocity (for the wall-normal stress). The final step is to couple this back to the LES equations (3) which can be done by using $\tau_w$ as a boundary condition for the LES momentum equation or by modifying the viscosity at the wall, $\nu_{BC}$ such that 

$$\tau_w = u^2 = \nu \left( \frac{\partial u}{\partial y} \right)_{y=0}$$

which can be inverted to give the value for the effective viscosity $\nu + \nu_{BC}$ at the wall, $\nu + \nu_{BC} = \frac{u^2 y_p}{v_p} = \frac{u^2 y_p}{v_p}$.

It is important to note the ‘input-output’ character of the wall-model irrespectively of it being of one-equation or algebraic nature. The wall-model takes information from the LES in the form of instantaneous data at grid points some distance above the wall and returns the wall shear stress or friction velocity to the LES at the wall, $y=0$. This data is then used by the LES to construct approximate boundary conditions at the wall, $y=0$.

Figure 4 attempts to illustrate the principal differences between wall-resolved LES, the two main branches of wall-modeled LES, algebraic and one-equation, and hybrid RANS-LES. For wall-resolved LES the main challenge is to create a computational grid that is sufficiently fine everywhere to resolve the Taylor scales, $\ell_T = \sqrt{\frac{\varepsilon}{\nu}}$, in which $\varepsilon$ is the dissipation rate and $\nu = \kappa^{1/2}$ the velocity fluctuations. When the wall is approached the Taylor scales gradually decrease in size until they approach the Kolmogorov scales, $\ell_K = \left( \frac{\nu^3}{\varepsilon} \right)^{1/4}$, which are the smallest scales in turbulence. From the definition of the Taylor and Kolmogorov scales if follows that $\ell_T = \ell_T/\text{Re}^{1/2}$ and $\ell_K = \ell_K/\text{Re}^{3/4}$, which in turn implies that $\ell_T/\ell_K = \text{Re}^{1/4}$, from which we may conclude that the characteristic scales become highly anisotropic, with aspect ratios of between 50 and 200 for high Re number flows, as the wall is approached. DNS results and experimental data indeed reveal that the near-wall flow is highly anisotropic, being dominated by large-scale high- and low-speed streaks aligned with the flow, being $\sim 1000(\nu/u)$ long, and having a spacing of $\sim 100(\nu/u)$, in which $(\nu/u)$ is the viscous length-scale in the boundary layer. Around these streaks we have vortex structures that are aligned with the streaks for some distance until they rise to form part of $\Omega$-shaped structures of different sizes. High-speed fluid, sweeps, moves from the outer part of the boundary layer into the inner part of the boundary layer and, conversely, low-speed fluid, ejections, moves from the inner to the outer part of the boundary layer. A wall-resolved LES have no problems capturing these features as showed for example for a turbulent pipe flow at $\text{Re}_\tau = 1000$, [64]. Here, a grid of 589 Mcells was required for the DNS whereas a grid of only 36 Mcells was required for the wall-resolved LES. The difference

![Figure 4](image-url)

**Figure 4**: The principles of (a) wall-resolved LES, (b) wall-modeled LES using algebraic wall-models (left) and one-equation wall-models (right) and (c) hybrid RANS-LES. The background is from wall-resolved, wall-modeled and hybrid RANS-LES simulations of fully developed turbulent channel flow at $\text{Re}_\tau = 1000$. The different grid systems are superimposed on the LES velocity predictions.

in skin friction between the DNS and LES predictions in comparison to hot-wire anemometry data were 1.7% and 5.2%, respectively, whereas the differences in boundary layer and momentum thickness, and shape factor were smaller. For wall-resolved LES it may be advanta-
geous to use unstructured grids with hanging nodes as illustrated in figure 4a to concentrate the grid resolution to the inner part of the boundary layer.

The two main branches of wall-modeled LES are illustrated in the left and right panels of figure 4b, respectively. In both cases, the LES grid is comparatively coarse in the wall normal direction, which significantly reduces the overall computational cost, but relies on a model to represent the physics in the inner part of the boundary layer and to some extent also the physics in the outer part of the boundary layer. The fact that these models work with both resolved and unresolved flow physics data makes their interactions with the underlying LES flow more complicated. The remedy to this issue is to make sure that there is an overlap region in which both the resolved LES fields and the wall-model are active and exchange information. Regarding the grid resolution in wall-resolved LES we anticipate a conventional LES grid with cell sizes of the size of the Taylor scales, as often assumed to the most appropriate cell size for an LES, all the way to the wall, also in the wall normal direction. Since the boundary layer flow physics is generally described in terms of viscous length scales, \( \ell_V = \frac{\nu}{u_\tau} \), which can be converted to Taylor scales, \( \ell_T = \ell_V (\nu / u_\tau) \), the streaks (\( \sim 1000\ell_V \) long, having a spacing of \( \sim 100\ell_V \) and a radius of \( \sim 5\ell_V \)) are generally rather well resolved in a wall-resolved LES with a grid spacing of \( O(\ell_T) \). It is, however, the smallest scales and flow structures associated with the velocity gradient and the viscosity at the wall, residing in the viscous sub-layer and in the buffer layer, that need to be explicitly handled by the wall-model.

Regarding the hybrid RANS-LES illustrated in figure 4c the interface between the RANS and LES regimes is allowed to vary depending on how the flow develops. Regarding the grid, it is determined in principle by the implicit requirement of the RANS model, usually requesting that \( y^+ \leq 1 \), typically resulting in a grid that is extensively stretched towards the wall, resulting in flat, sheet-like grid cells at the wall. Most hybrid RANS-LES models suffer from some degree of problem in maintaining the correct mean velocity profile around the interface. Most often, the mean velocity \( \bar{v}(y^+) \) in the LES region ends up above the log-law, and thus this is called the “log-layer mismatch”. Multiple studies have reported improvements that can remove the log-layer mismatch, through either the addition of small-scale forcing or by tailoring the blending function between the RANS and LES eddy-viscosities. The caveat, however, is that the results depend quite strongly on the forcing amplitude, [65].

4 CANONICAL FLOW CASES

In order understand more about the RANS, DES and LES models we next summarize the application of these models to a few relevant canonical flow cases.

4.1 Fully Developed Turbulent Channel Flow

Fully developed turbulent channel flow is a simple canonical building block flow that has been examined using RANS, DES, LES and DNS for decades, e.g. [58, 66-69]. The channel is typically confined between two parallel plates, 2h apart, where h is the channel half-width. The flow is driven by a mass flow in the axial (e_x) direction, no-slip conditions are applied in the cross-stream (e_y) direction, and periodic conditions are applied in the spanwise (e_z) direction. Here, results will be discussed for Re_τ=550 and 1000, 2000 and 5200, following the DNS of Lee & Moser, [58]. The channel used is 9h × 2h × 4h in the axial, wall-normal and spanwise directions, respectively. For wall-resolved LES the grid is designed with \( \Delta x = \Delta y = \Delta z = 0.0333 \cdot h \) in the core of the channel, whereas in the near-wall region, \( y < 0.0333 \cdot h \), the grid is uniformly
refined in all three directions so that \( y^+ \approx 1 \). For the different Re numbers, this strategy results in grids of 4.21, 11.47, 40.50 and 152.5 Mcells, respectively. For wall-modeled LES the grid is designed so that \( \Delta x = \Delta y = \Delta z = 0.0333 \cdot h \) in the whole channel, resulting in 1.94 Mcells, and \( y^+ \) values of 9, 16, 32 and 84, respectively, for the different Re numbers. For DES, the grid is designed with \( \Delta x = \Delta y = \Delta z = 0.0333 \cdot h \) in the core of the channel, whereas in the near-wall region, \( y < 0.0333 \cdot h \), stretching is applied in the wall normal direction so that \( y^+ \approx 1 \) at the wall, resulting in between 2.65 and 3.88 Mcells depending on Re.

The upper and lower panels of figure 6a show perspective views of the flow in the channel from wall-modeled LES using the LDKM model, [15], and wall-resolved LES using the WA-LE model, [15], at Re\(_{\tau}=2000\), in terms of contours of the axial velocity, \( v_x \), and the friction velocity, \( u_\tau \), at the lower wall, and iso-surfaces of the second invariant, \( \lambda_2 \), of the velocity gradient tensor, \( \nabla v \). For both the wall-resolved and wall-modeled LES the \( \lambda_2 \)-iso-surfaces reveal a plethora of different vortical structures: Streamwise vorticies, forming an acute angle with the wall, are observed to agglomerate in the low-speed streaks, and hairpin vorticies, rapidly bending away from the wall, form due to shear between the low-speed and high-speed streaks. Thinner, mainly streamwise, vortical structures are located in the wall-resolved LES between and beneath the low-speed and high-speed streaks. The streaks are \( \sim 1000 \cdot \nu/u_\tau \) long, having a spacing of \( \sim 100 \cdot \nu/u_\tau \), and the hairpin vortices have a diameter of between 2 and \( 10 \cdot \nu/u_\tau \). The wall-modeled LES cannot resolve the vortical structures between and beneath the streaks, but resolves the streaks as well as the largest (and strongest) hairpin vortices.

Figure 5: Fully developed turbulent channel flow. In (a) results from wall-modeled (top) and wall-resolved (bottom) LES at Re\(_{\tau}=2000\) are presented in terms of \( \lambda_2 \). In (b) and (c) profiles of the time-averaged axial velocity, \( \langle v_x \rangle \), and its rms fluctuations, \( v_x^{\text{rms}} \), are presented for Re\(_{\tau}=550\), 1000, 2000 and 5200, respectively. Legend: (---) DNS, [58], (---) wall-resolved LES using the LDKM model, (---) wall-modeled LES using the WA-LE model, (---) wall-modeled LES using the LDKM model, (---) hybrid RANS-LES using the DES model, [14], and (---) hybrid RANS-LES using the IDDES model, [26-27].

Figures 5b and 5c compare the mean streamwise velocity, \( \langle v_x \rangle \), and the streamwise rms-velocity fluctuations, \( v_x^{\text{rms}} = \sqrt{\langle (v_x - \langle v_x \rangle)^2 \rangle} \), normalized with \( u_\tau \), from DNS, [58], wall-resolved LES, using the WA-LE subgrid models, wall-modeled LES, using the WA-LE and LDKM subgrid models, DES, [14], and IDDES, [26-27]. For \( \langle v_x \rangle \) we find excellent agreement between DNS and wall-resolved LES for Re\(_{\tau}=550\), 1000 and 2000, with no existing LES for Re\(_{\tau}=5200\). The wall-modeled LES present very good agreement with the DNS given the coarse grids in the near-wall region. Virtually no difference is noted between the WA-LE and LDKM subgrid models. For DES and IDDES we find that IDDES performs better than DES, and both models
show good agreement across the whole range of Re numbers. Evidence of the ‘log-layer mismatch’ can however be detected in the DES, IDDES and wall-modeled LES. For $v_{rms}^w$ we find good agreement between the DNS and the wall-resolved LES, whereas the wall-modeled LES predicts a peak in $v_{rms}^w$ at the first grid point instead of at $y'\approx 12$ due to the coarse grid. Apart from that, the agreement is satisfactory. For the DES and IDDES results we find that both of these methods fail to predict $v_{rms}^w$ in the core flow, and IDDES also underpredicts the peaks in $v_{rms}^w$. For the wall-resolved LES the error in skin-friction, $C_f$, is $\sim 2\%$ at $Re=5200$. For the wall-modeled LES, DES and IDDES the error in $C_f$ is between 5\% and 9\%.

4.2 Zero Pressure Gradient Flat Plate Boundary Layer Flow

The next level of complexity is facilitated by the flow over a flat plate at zero pressure gradient. Historically, knowledge of flat plate turbulent boundary layers was gained experimentally, [70-72], but more recently have DNS, [73-74], and LES, [75-76], also significantly contributed to the current level of understanding of boundary layers. Spatially developing boundary layer simulations are typically set up in a rectilinear computational domain as illustrated in figure 6a. Here, a computational domain is set-up to emulate the experimental study of DeGraaf & Eaton, [72], and the DNS of Schlatter & Örlü, [74]. The domain is 2.00 m long, 0.20 m wide and 0.127 m high, and is discretized with hexahedral cells having cell sizes of $\Delta x=2.6$ mm, $\Delta y=0.80$ mm and $\Delta z=2.00$ mm, resulting in a baseline grid of 12.16 Mcells. A uniformly refined grid with 97.28 million cells has been used to study the statistical grid convergence. No-slip boundary conditions are applied at the wall, periodic boundary conditions are applied in the spanwise direction, whereas open inflow/outflow boundary conditions are applied at the inlet and outlet, respectively. RANS, with the SST $k-\omega$ model, wall-modeled LES, with the WALE and LDKM models, and hybrid RANS-LES, with the DES and IDDES models, have here been performed. The LES are provided with a numerical trip, 20 mm downstream of the inlet, at $Re=100$, to stimulate the boundary layer development.

In figure 6a we present a few selected views of the flat plate boundary layer flow in terms of $k_z$, instantaneous and time-averaged velocity contours, $\bar{v}_x$ and $\langle v_x \rangle$, and friction velocity, $u_\tau$, contours, and in (b) and (c) normalized axial velocity profiles, $\langle v_x \rangle$, are compared for $Re\approx 2900$ and 5200, respectively. Legend: ($\times$) experimental data, [72], ($\longrightarrow$) DNS, [74], ($\longrightarrow$) wall-modeled LES using the WALE model, ($\longrightarrow$) wall-modeled LES using the LDKM model, ($\longrightarrow$) hybrid RANS-LES using the DES model, ($\longrightarrow$) hybrid RANS-LES using the IDDES model, and ($\longrightarrow$) RANS using the SST $k-\omega$ model.

Figure 6: Zero Pressure Gradient Flat Plate Boundary Layer flow. In (a) results from wall-modeled LES are presented in terms of $k_z$, instantaneous and time-averaged velocity contours, $\bar{v}_x$, and $\langle v_x \rangle$, and friction velocity, $u_\tau$, contours, and in (b) and (c) normalized axial velocity profiles, $\langle v_x \rangle$, are compared for $Re\approx 2900$ and 5200, respectively. Legend: ($\times$) experimental data, [72], ($\longrightarrow$) DNS, [74], ($\longrightarrow$) wall-modeled LES using the WALE model, ($\longrightarrow$) wall-modeled LES using the LDKM model, ($\longrightarrow$) hybrid RANS-LES using the DES model, ($\longrightarrow$) hybrid RANS-LES using the IDDES model, and ($\longrightarrow$) RANS using the SST $k-\omega$ model.
of instantaneous and time-averaged axial velocities, $v_x$, and $\langle v_x \rangle$, wall-friction velocities, $u_\tau$, and iso-surfaces of the second invariant of the velocity gradient tensor, $\nabla v$, from the wall-modeled LES with the LDKM model on the baseline grid. All simulations result in boundary layers that thicken with downstream distance from the leading edge approximately according to $\delta \approx 0.16x/Re_\theta^{1/7} = 0.16x^{8/7}(v/v_0)^{1/7}$, [70]. The boundary layers predicted by the LES models are all populated by a plethora of vortices, developing downstream of the numerical trip, whereas those predicted by the DES and IDDES seem virtually free of vortical structures. In the LES, the disturbances created by the numerical trips gradually develop into coherent vortical structures in which the streamwise vorticies, forming an acute angle with the wall, agglomerate in the low-speed streaks, and hairpin vorticies, rapidly bending away from the wall, form due to shear between the low-speed and high-speed streaks. The vortices resolved in the wall-modeled LES are typically thicker than those resolved in the target DNS, but reveal the same topology and dynamic behavior as the target DNS. This behavior is similar to the so-called ‘fat worms’ observed in LES of homogeneous isotropic turbulence, [77].

Figures 6b and 6c present time-averaged axial velocity profiles, $\langle v_x \rangle$, normalized by the freestream velocity, $v_0$, at two different locations corresponding to momentum thickness Re-numbers of $Re_\theta \approx 3030$ and 4060, respectively. The boundary layer thickens with downstream distance from the leading edge, as observed in the experimental data from deGraaff & Eaton, [72], and from the DNS of Schlatter & Örlu, [74], and it is apparent that the RANS SST-kω prediction captures this behavior well. The DES model underpredicts the growth rate of the boundary layer, and predicts a more laminar like profile along the whole flat plate. The IDDES models predict a boundary layer in agreement with the experimental and DNS data. The wall-resolved LES, using the WALE and LDKM subgrid models, respectively, show virtually indistinguishable boundary layer profiles in reasonable agreement with the experimental and DNS data, but with a somewhat broader and fuller $\langle v_x \rangle$-profile.

5 INTERMEDIATE COMPLEXITY FLOW CASES

To further enhance the understanding of the RANS, DES and LES models we summarize the application of these models to a few selected flow cases of higher geometrical and induced physical complexity relevant to marine or offshore interest.

5.1 Flow around a 3D Bump

The next case considered is the flow over a 3D hill attached to a rectilinear wind tunnel section, [78-80]. The experiment feature a $h=0.078$ m high 3D axisymmetric hill mounted on the floor of a $H=0.25$ m high wind tunnel. The inlet speed is $v_0=27.5$ m/s giving a Re number, based on $h$, of $Re \approx 143,000$. The wind tunnel test section is 7.62 m long and 3.03 m wide, and the hill is analytically defined, [78]. LES of this flow have been performed by e.g. Patel & Menon, [81], Persson et al., [82], Visbal & Risetta, [83], and Garcia-Villalba et al., [84], and RANS have been performed by several researchers as summarized in [85]. Different computational domains are used by different investigators but as illustrated in figure 7a two approaches prevail: short domains using pre-cursor channel flow simulations, [84], and simulations of the whole test section of the wind tunnel, [82], to model the boundary layer evolution. The grids used in [81-85] range from 4.0 to 134.5 Mcells, and the objectives of these computations are also different as explained in the references. Here, we will compare the wall-resolved LES of Garcia-Villalba et al., [84], with wall-modeled LES, using the LDKM and WALE subgrid
models, and RANS using the SST-k-ω model. Open inflow/outflow boundary conditions are used at the in- and outlets, respectively, no-slip boundary conditions are used at the upper and lower channel walls and slip conditions are applied in the spanwise direction.

The flow over and around the 3D hill is very complicated as explained in all the references cited, and as also seen from figure 7b, showing instantaneous iso-surfaces of the second invariant of the velocity gradient tensor, $\lambda_2$, colored by the velocity, $v_x$. As the boundary layer approaches the 3D hill, the pressure increases but the increase is not large enough to cause separation. As the flow accelerates over the top of the hill the pressure decreases. Minimum pressure occurs at the top, and is followed by an adverse pressure gradient at the lee-side of the hill, which results in complex flow separation and reattachment in the shallow wake of the hill. Oil-flow visualization and skin-friction visualizations from the LES shows that the separation and reattachment is very complicated with multiple unsteady vortical structures. These flow structures result as a consequence of the acceleration over the top and around the sides of the hill due to the favorable pressure gradients in both directions. The spanwise favorable pressure gradient in front of the hill causes the flow to diverge outward, however, at the lee-side this gradient becomes adverse, causing the side boundary layers to converge on the back of the hill and closer to the center plane in a high-pressure region. The presence of the 3D hill modifies the vorticity distribution, primarily through the pressure gradients, and results in a wake that is inhabited by hairpin-vortices that are deformed by the high level of resolved and subgrid turbulence in the wake behind the 3D hill.

Figure 7: Flow around a 3D bump. In (a) different computational set-ups are presented, (b) results from wall-modeled LES in terms of $\lambda_2$, and in (c), (d) and (e) normalized velocity and turbulence profiles, $\langle v_x \rangle$, $\langle v_z \rangle$ and $k$, respectively, are compared. Legend: (×) experimental data, [79], (○) experimental data, [80], (---) wall-resolved LES using the DSMG model, [84], (-----) wall-modeled LES using the WALE model, (----) wall-modeled LES using the LDKM model and (==) RANS using the SST-k-ω model.

Figures 7c to 7e show normalized and time-averaged streamwise, $\langle v_x \rangle$, and cross-stream,
velocity profiles, together with profiles of the turbulent kinetic energy, $k$. Experimental
data from LDV of Byun & Simpson, [79], and hot-wire of Ma & Simpson, [80], are included
together with wall-resolved LES predictions of Garcia-Villalba et al., [84], and wall-modeled
LES predictions, using the LDKM and WALE subgrid models, together with RANS predic-
tions, using the SST $k$-$\omega$ model. The first observation is the similarity of the two experimental
velocity data sets, and the difference in the corresponding $k$ data sets. This is due to the differ-
ence in measurement technique employed, and highlight the complexity of performing accu-
rate flow measurements of complex cases. The RANS predictions do not capture the essential
features of the flow, resulting in poor predictions of both $\langle v_x \rangle$, $\langle v_z \rangle$ and $k$. The RANS results
are good representatives of what may be expected from modern RANS as may be observed by
comparing also with other RANS results in [85]. The wall-resolved LES, [84], show extreme-
ly good agreement with both experimental velocity data sets, and with the hot-wire experi-
mental data set for $k$, supporting the use of hot-wire anemometry. The two wall-modeled LES
predictions show very similar results, with only minor variations due to choice of subgrid mo-
del, that also agrees favorably with the wall-resolved LES, [84], and both experimental veloc-
ity data sets, and with the hot-wire experimental data set for $k$.

5.2 Flow around a Generic Bare Hull Submarine Configuration

The final case considered is the flow around the bare-hull version of the DSTG Joubert ge-
neric submarine model created from the work of Joubert, [45-46], and discussed in more de-
tail in [47]. The model have an overall length of 1.35 m, a length to diameter ratio of 7.3, and
consists of a cylindrical mid-body, an ellipsoid bow and a parabolic stern, figure 8a. Represent-
ative of modern submarine shapes, this generic test article has no full-scale equivalent.
The model was experimentally tested in the DSTO low speed wind tunnel, [86], in the octag-
onal test section of which it was mounted using a floor-mounted pylon with a shrouded fair-
ing. The experimental testing was boundary layer tripping devices, which consisted of either a
circular wire of diameter 0.2 mm or 0.5 mm, or a 3 mm wide circumferential strip of distrib-
uted silicon carbide grit of size 80, [47]. The trip was located at an axial coordinate of $x=67.5$
mm measured from the nose of the hull, which corresponds to $x/L \approx 0.05$. Moreover, the loca-
tion of the trip was positioned far enough upstream to ensure it did not experience an adverse
pressure gradient, which under such conditions could lead to undesirable separation, and far
enough downstream to ensure the boundary layer would not relaminarize. The overall block-
age ratio for the model at zero-incidence was estimated to be about 2.2%. The model was
equipped with flush mounted static pressure tappings along the top centerline of the hull in
order to estimate the mean pressure coefficient $C_p$, whereas the mean turbulent skin-friction
coefficients, $C_{f_t}$ was measured using the Preston tube method, [87]. The velocity around the
upper part of the stern section was measured with Particale Imaging Velocimetry (PIV) in two
patches shown in figure 8a. These two patches are stitched together to provide a complete de-
scription of the stern flow. Some influence of the mounting was observed. The experimental
Re number, based on the hull length and freestream velocity was $5.4 \times 10^6$.

Simulations of this case with RANS, using the SST-$k$-$\omega$ model, hybrid RANS-LES, using
the DES and IDDES models, and wall-modeled LES, using the WALE and LDKM models,
have been performed. Hexahedral grids with 17.0 Mcells, [47], and unstructured grids with 44
and 126 Mcells have been used. No-slip boundary conditions are applied on the hull, whereas
conventional freestream and open inflow-outflow boundary conditions are used at the outer
computational boundaries. The freestream turbulence was estimated to 3% in the RANS, DES and IDDES as measured in the wind-tunnel, whereas in LES no freestream turbulence was included. In the IDDES and LES, a numerical trip model was included to model the effects of the trip wire or trip grit used in the wind-tunnel experiments.

Figure 6b shows the vorticity distribution over the hull in terms of iso-surfaces of the second invariant of the velocity gradient tensor, \( \nabla v \), denoted \( \lambda_2 \), from the wall-modeled LES using the LDKM model. The iso-surfaces of \( \lambda_2 \) clearly the presence of the numerical trip, before which the boundary layer is laminar, and after which a turbulent boundary layer gradually develops. The boundary layer remains turbulent along the whole parallel midsection of the hull, and towards the tapered stern an adverse pressure gradient assists in thickening the boundary layer. The LES models as well as the RANS models predict an attached boundary layer all the way to the stern tip. The \( \lambda_2 \) structures of which the boundary layer is composed consists initially of azimuthal disturbances that gradually breaks up to develop a carpet of hairpin-type vortices, the legs of which forms an acute angle to the hull, whereas the neck rises sharply as it bends sideways to connect with is companion forming on the other side of the high-speed or low-speed streak. The hairpin vortex elements are thicker than anticipated, but they behave in the expected way, and so this behavior can be similar to the so-called ‘fat worms’ observed in the spatially developing boundary layer of Section 4.2, and also in early LES of homogeneous isotropic turbulence as described by Ashurst et al., [77]. Along the tapered stern cone, in the thickening boundary layer, the cross-wise momentum transfer creates streamwise vortices that are transported down behind the hull and into the gradually developing wake.

Figure 8: Flow around the DSTG Joubert generic bare-hull submarine configuration. In (a) the hull-shape is outlined together with the PIV patches. In (b) results from wall-modeled LES, using the LDKM model, is presented in terms of \( \lambda_2 \) colored by the streamwise velocity \( v_x \), and in (c) and (d), comparisons of \( C_P \) and \( \langle v_x \rangle \) are presented. Legend: \( \times \) experimental data, [47], \( \textcolor{teal}{
\text{-}} \) wall-modeled LES using the WALE model, \( \textcolor{indigo}{
\text{-}} \) wall-modeled LES using the LDKM model, \( \textcolor{blue}{
\text{-}} \) hybrid RANS-LES using the DES model, \( \textcolor{orange}{
\text{-}} \) hybrid RANS-LES using the IDDES model, and \( \textcolor{green}{\text{-}} \) RANS using the SST \( k\-\omega \) model.

Figure 6c compares the time-averaged or mean static pressure coefficient, \( C_P \). The agreement between the experimental \( C_P \) profile and the computed profiles from the RANS, hybrid RANS-LES and LES predictions is very good along the whole hull. Note that the \( C_P \) profile from the SST-\( k\-\omega \) RANS model is hidden beneath the other profiles. Virtually no difference at all can be observed between the two wall-modelled LES predictions, whereas some minor
C. Fureby

6 CONCLUDING REMARKS

High-fidelity (LES and hybrid RANS-LES) numerical simulations of ship and submarine hydrodynamics have reached a certain level of maturity, yet there are still obstacles to overcome before these methods become an everyday tool in marine engineering. The central challenge is the very high Re number encountered in ship- and submarine hydrodynamics, and how to manage the associated issue of an extensive range of eddy scales, ranging from the size of the large eddies of the hull to the smallest Kolmogorov scales. The eddy scales associated with the hull can be O(1) m whereas the associated Kolmogorov scales, representing dissipation of turbulence into heat, can be O(10) mm. The intermediate Taylor scales, at which viscosity affects the turbulence dynamics, can be O(1) mm. This extreme range of scales makes DNS unfeasible for both full-scale and model scale applications. The alternative of RANS have been successfully used for decades to compute the mean flow around numerous hull forms, and is also widely used in aerodynamics and other related fields. The major drawbacks of RANS are that it cannot handle large-scale unsteadiness, and that the method is not designed to provide information about unsteady features of the flows or small details of the flows. This essentially leaves the two branches of wall-resolved and wall-modeled LES, as well as hybrid RANS-LES, as the main candidates for practical ship and submarine flow simulations. These methods are typically required when accurate predictions are needed that also include information about dynamic features such as pressure and turbulence fluctuations that may influence flow noise and vibrations. Wall-resolved LES is however found to be too expensive for today’s computational architectures with respect to full-scale predictions, whereas predications of model-scale hull forms are now just manageable.

This leaves wall-modeled LES and hybrid RANS-LES as the only currently feasible candidates for high-fidelity ship and submarine hydrodynamics. The use of any of these methods is not entirely straightforward, and requires substantial knowledge about the methodology as well as some background information about the flow to be computed. This background information can be obtained by RANS and is usually employed to generate computational grids well suited for the two methods of wall-modeled LES and hybrid RANS-LES. Different wall-modeled LES and hybrid RANS-LES models are available but the underlying principles are the same irrespectively of which model is used. Hybrid RANS-LES requires a computational
grid that switches, seamlessly, from an LES grid in the detached flow to a RANS grid in the attached flow. Wall-modeled LES requires an LES grid throughout the whole computational domain. Hybrid RANS-LES relay on an interface, explicit or implicit, to determine when to switch from LES to RANS, whereas wall-modeled LES is based on solving the LES equations all the way to the wall. A wall-model is used just adjacent to the wall to estimate the wall-shear stress (or the friction velocity) from the LES solution in order to provide a boundary condition for the wall-modeled LES.

The flow predictions used to illustrate different aspects of ship and submarine flow physics clearly points at some lack of understanding of high Re number turbulent boundary layers developing over curved and complex surfaces. More high-quality experimental data as well as DNS simulation results are needed to fill the gaps in our current understanding of the boundary layer flow physics. Not until this gap has decreased, improved simulation models, based on the underlying physics can be developed. It is anticipated that such simulation models will be of multi-scale nature, [88], to reflect the multi-scale nature of the boundary layer. The continuous development of algorithms and hardware will help in rapidly reaching higher and thus more relevant Re number flows which are likely to reveal new physical challenges in terms of new physical mechanisms for the composition of a turbulent boundary layer.

ACKNOWLEDGEMENT

The author acknowledges the generous supply of simulation results and many valuable discussions and comments from Brendon Andersson, Daniel Norrison, William Sidebottom, David Clarke, David Jones, Matteo Giacobello, Peter Manovski, Simon Henbest, Ivan Marusic, Myoungkyu Lee, Magnus Tormalm, Suresh Menon, Serge Toxopeus, Fernando Grinstein, Rickard Bensow, Mattias Johansson, Kristian Petterson and Mattias Liefvendal.

REFERENCES

[34] Xing T., Bhushan S. & Stern F.; 2012, “Vortical and Turbulent Structures for KVLOC2 at Drift angle 0, 12, and 30 Degrees”, Ocean Engineering, 55, p 23.


ARE RANDOM COEFFICIENTS NEEDED IN PARTICLE SWARM OPTIMIZATION FOR SIMULATION-BASED SHIP DESIGN?

ANDREA SERANI AND MATTEO DIEZ

CNR-INSEAN, National Research Council-Marine Technology Research Institute
Via di Vallerano 139, 00128 Rome, Italy
e-mail: andrea.serani@insean.cnr.it, matteo.diez@cnr.it

Key words: Global derivative-free optimization, Particle swarm optimization, Stochastic versus deterministic optimization, Simulation-based design, Ship hydrodynamics

Abstract. Simulation-based design optimization (SBDO) methods integrate computer simulations, design modification tools, and optimization algorithms. In hydrodynamic applications, often objective functions are computationally expensive and likely noisy, their derivatives are not directly provided, and the existence of local minima cannot be excluded a priori, which motivates the use of derivative-free global optimization algorithms. This type of algorithms (such as Particle Swarm Optimization, PSO) usually follow a stochastic formulation, requiring computationally expensive numerical experiments in order to provide statistically significant results. The objective of the present work is to investigate the effects of using (versus suppressing) random coefficients in PSO for ship hydrodynamics SBDO. A comparison is shown of 1,000 random PSO to deterministic PSO (DPSO) using 12 well-known scalable test problems, with dimensionality ranging from two to fifty. A total of 588 test functions is considered and more than 500,000 optimization runs are performed and evaluated. The results are discussed based on the probability of success of random PSO versus DPSO. Finally, a comparison of random PSO to DPSO is shown for the hull-form optimization of the DTMB 5415 model. In summary, test functions show the robustness of DPSO, which outperforms random PSO with odds of 30/1 for low-dimensional problems (indicatively $N \leq 30$) and 5/1 for high-dimensional problems ($N > 30$). The hull-form SBDO ($N = 11$) shows how DPSO outperforms PSO with odds of 20/1. The use of DPSO in the SBDO context is therefore advised, especially if computationally expensive analyses are involved in the optimization.

1 INTRODUCTION

The design of marine vehicles requires accurate analyses and complex decision-making methodologies. In the last decades, the design paradigm has shifted from the build-and-test approach to more efficient and versatile simulation-based methodologies. The integration of optimization algorithms with computer simulations has led to automatic simulation-based design optimization (SBDO) procedures, with the aim of guiding the designer in the decision making process.
of complex engineering applications. For shape optimization problems, SBDO consists of three main elements: (i) a geometry modification and automatic meshing tool, (ii) a simulation tool, and (iii) an optimization algorithm, which need to be integrated in an efficient and robust way.

Among numerous types and implementations of optimization algorithms, global derivative-free methods often represent a robust option for SBDO. The success of this type of methods stems from the peculiar characteristics of SBDO (in ship hydrodynamics as well as other engineering fields). The optimization objectives are often the results of complex simulations, solving systems of partial differential equations. These results are often noisy, due to the presence of solution residuals. Access to the source code is not always possible, therefore derivative-based methods (whether used) need to rely on finite differences, which may be highly inaccurate. Furthermore, in most problems the existence of multiple local optima cannot be excluded a priori, making the use of global derivative-free methods very attractive.

Among global derivative-free optimization algorithms, Particle Swarm Optimization (PSO) [1] has gained the attention of the marine engineering community, due to the ease of implementation and capability of providing approximate solutions to the optimization problem at a reasonable computational cost. PSO is based on the social-behaviour metaphor of a flock of birds or a swarm of bees searching for food. PSO belongs to the class of heuristic algorithms for single-objective evolutionary optimization. Recent applications of PSO to ship SBDO include hull-form and waterjet design optimization of fast catamarans [2, 3, 4], the optimization of unconventional multi-hull configurations [5], and surface combatant [6, 7].

The original PSO formulation uses random coefficients to sustain the swarm dynamics. This property implies that statistically significant results can be obtained only through extensive numerical campaigns. Such an approach can be too expensive in SBDO for complex industrial applications, where CPU-time expensive computer simulations are used directly as analysis tools. For these reasons, efficient deterministic approaches (such as deterministic PSO, DPSO) have been developed and investigated [8, 9], suppressing any kind of randomness in the particle position update. In this deterministic version, the swarm diversity is sustained only by the swarm dynamics. During the swarm evolution each particle is attracted by diverse positions, based on the cognitive and social experience, iteration by iteration. In most problems, this is generally sufficient to maintain the swarm sufficiently active and provide reasonable solutions.

The objective of the present work is to investigate the effects of using (versus suppressing) random coefficients of PSO in SBDO for ship hydrodynamics.

The approach includes a comparison of 1,000 random PSO to DPSO, using 12 well-known analytical test functions, with dimensionality ranging from two to fifty. The total number of functions assessed is 588 with a total number of optimization runs larger than 500,000. The numerical results are discussed based on the probability of success of random PSO versus DPSO. Finally, the comparison is shown for the hydrodynamic hull-form optimization of the DTMB 5415 model, an early and open to public version of the DDG-51 destroyer. A single-speed single-objective example is shown, aimed at the reduction of the total resistance coefficient in calm water at 18 kn, corresponding to Froude number (Fr) equal to 0.25. The design constraints include fixed displacement and length between perpendiculars, along with a ±5% maximum variation of beam and draft. The ship is free to sink and trim. An expansion of eleven orthogonal basis functions [10] is used for the modification of the hull form and the sonar dome. The solution of the problem is based on a metamodel [11], trained by a RANS solver (CFDShip-Iowa v4.5
2 GENERAL FORMULATION OF THE GLOBAL OPTIMIZATION PROBLEM

Consider the following objective function

\[ f(x) : \mathbb{R}^N \rightarrow \mathbb{R} \]  \hspace{1cm} (1)

and the global optimization problem

\[ \min_{x \in \mathcal{L}} f(x), \quad \mathcal{L} \subset \mathbb{R}^N \]  \hspace{1cm} (2)

where \( \mathcal{L} \) is a closed and bounded subset of \( \mathbb{R}^N \) and \( N \) is the number of variables, collected in \( x \). The global minimization of the objective function \( f(x) \) requires to find a vector \( a \in \mathcal{L} \) such that:

\[ \forall b \in \mathcal{L} : \quad f(a) \leq f(b) \]  \hspace{1cm} (3)

Then, \( a \) is a global minimum for the function \( f(x) \) over \( \mathcal{L} \). Hereafter, the compact set \( \mathcal{L} \) represents box constraints.

Since the solution of Eq. 2 is in general an NP-hard problem, the exact identification of a global minimum might be very difficult. Therefore, solutions with sufficient good fitness, provided by heuristic procedures (such as PSO), are often considered acceptable for engineering purposes.

3 PARTICLE SWARM OPTIMIZATION

3.1 Stochastic formulation

The original formulation of the PSO algorithm, as presented in [13], reads

\[
\begin{align*}
    v_{i}^{k+1} &= w v_i^k + c_1 r_{1,i}^k (p_i - x_i^k) + c_2 r_{2,i}^k (g - x_i^k) \\
    x_{i}^{k+1} &= x_i^k + v_i^{k+1}
\end{align*}
\]  \hspace{1cm} (4)

The above equations update velocity \( (v_i^k) \) and position \( (x_i^k) \) of the \( i \)-th particle at the \( k \)-th iteration, where: \( w \) is the inertia weight; \( c_1 \) and \( c_2 \) are respectively the social and cognitive learning rate; \( r_{1,i}^k \) and \( r_{2,i}^k \) are uniformly distributed random numbers in \([0, 1] \); \( p_i \) is the personal best position ever found by the \( i \)-th particle in the previous iterations and \( g \) is the global best position ever found in the previous iterations by all the particles.

An overall constriction factor \( \chi \) may be used [14], in place of the inertia weight \( w \). Accordingly, the system in Eq. 4 is recast in the following equivalent form

\[
\begin{align*}
    v_{i}^{k+1} &= \chi \left[ v_i^k + c_1 r_{1,i}^k (p_i - x_i^k) + c_2 r_{2,i}^k (g - x_i^k) \right] \\
    x_{i}^{k+1} &= x_i^k + v_i^{k+1}
\end{align*}
\]  \hspace{1cm} (5)
3.2 Deterministic formulation

A deterministic version of the PSO algorithm (namely DPSO) was formulated in [8] by setting \( r_{1,i}^k = r_{2,i}^k = 1, \forall i, k \) in Eq. 5

\[
\begin{align*}
    v_{i}^{k+1} &= \chi \left[ v_{i}^{k} + c_1 (p_{i} - x_{i}^{k}) + c_2 (g - x_{i}^{k}) \right] \\
    x_{i}^{k+1} &= x_{i}^{k} + v_{i}^{k+1}
\end{align*}
\]  

(6)

A discussion for an effective and efficient use of DPSO for SBDO in ship hydrodynamics has been presented in [9].

3.3 Parameter setup

The parameter setup used for both random PSO and DPSO is selected as suggested in [9]: number of particles \( N_p = 4N \); particle initialization with Hammersley sequence sampling distribution on domain and bounds (for \( N < 10 \)) and domain only (for \( N \geq 10 \)) with non-null velocity [3]; set of coefficients proposed in [15], i.e., \( \chi = 0.721, c_1 = c_2 = 1.655 \); semi-elastic wall-type approach [9]. A limit to the number of objective function evaluations is set equal to 400N, corresponding to 100 algorithm iterations.

4 TEST PROBLEMS

4.1 Analytical test functions

Twelve analytical test problems are used in the preliminary numerical experience, including a wide variety of functions, such as continuous and discontinuous, differentiable and non-differentiable, separable and non-separable, unimodal and multimodal. Each problem is studied with dimensionality \( 2 \leq N \leq 50 \), resulting in a total number of test functions equal to 588. Tables 1 summarizes the test problems used in the current study, including variable bounds and global optimum.

4.2 Hull-form optimization of the DTMB 5415

![Figure 1: A replica of the DTMB 5415 (CNR-INSEAN model 2340)](image)

The hull-form optimization of the DTMB 5415 is used to assess the algorithm performance for ship hydrodynamic problems. Figure 1 shows the geometry of a 5.720 m length DTMB 5415 model used for towing tank experiments, as seen at CNR-INSEAN [16]. The optimization is formulated as

\[
\begin{align*}
    &\text{Minimize} \quad f(x) \\
    \text{subject to} \quad &g_l(x) = 0, \quad \text{with} \quad l = 1, \ldots, L \\
    \text{and to} \quad &h_m(x) \leq 0, \quad \text{with} \quad m = 1, \ldots, M
\end{align*}
\]  

(7)
In the current study, a limit to the number of function evaluations is set to 4400, i.e., 400N.
A. Serani and M. Diez

5 PERFORMANCE METRICS

5.1 Distance to the desired optimum

In order to assess the performance of both random PSO and DPSO, an overall distance-based metric, $\Delta_t$, is used [17]

$$\Delta_t = \sqrt{\frac{\Delta_x^2 + \Delta_f^2}{2}}$$

where

$$\Delta_x = \frac{1}{N} \sum_{j=1}^{N} \left( \frac{g_j - x_{j,\min}^*}{Z_j} \right)^2$$

and

$$\Delta_f = \frac{f(g) - f_{\min}^*}{f_{\max}^* - f_{\min}^*}$$

(8)

$\Delta_x$ is a normalized Euclidean distance between the global best found by the algorithm ($g$) and a desired (analytical if known, otherwise a known benchmark value) minimizer ($x_{\min}^*$), where $Z_j = |u_j - l_j|$ is the range of the $j$-th variable. $\Delta_f$ is the associated normalized difference (error) in the function space, where $f(g)$ is the minimum found by the algorithm, $f_{\min}^*$ is the analytical minimum, and $f_{\max}^*$ is the analytical maximum of the function $f(x)$ in the variable domain.

5.2 Probability of success of random PSO versus DPSO

The success probability (SP) of random PSO versus DPSO is defined as

$$SP = \frac{|S \cap T|}{|T|}$$

(9)

where $T$ is a random set of sets, each containing a number of $R$ solutions obtained by random PSO. Note that $T$ has dimension $|T|$ and contains sets of dimension $R$. $S$ is a subset of $T$ defined as

$$S = \{ s \in T \mid \text{argmin}[\Delta_t(s)] < \Delta_t(d) \}$$

(10)

with $\Delta_t(s) \in \mathbb{R}^R$ collecting $\Delta_t$ values from the set of $R$ random PSO solutions and $\Delta_t(d)$ indicating the solution obtained by DPSO.
SR indicates the probability for random PSO to find at least one solution better than DPSO. This is evaluated using a large number $|T|$ of different random sets and comparing them to the deterministic solution. SR generally depends on the number of variables, $N$, and the number of random PSO runs considered, $R$. Hereafter, $|T| = 1,000$.

6 RESULTS
6.1 Analytical test functions

Figure 3a shows the average $\Delta t$ (Eq. 8) of random PSO and DPSO, for problems of dimension $N$. For random PSO, the 95% confidence interval (CI) and the median is shown using 1,000 optimization runs. It can be seen how the performance of DPSO are fairly close to the median of random PSO and always falls within the 95% CI of random PSO. Both deterministic and random PSO show a more satisfactory performance for low numbers of variables.

Figure 3b shows the average success probability (SP) of random PSO versus DPSO (Eq. 9). The 95% probability indicates, for each $N$ and on average, the minimum number of random PSO runs required to obtain at least one solution better than DPSO. This number may vary significantly, depending on $N$. Generally, one random optimization does not guarantee a high probability of success. In order to achieve a success probability equal to 95%, at least five random PSO repetitions are needed, often significantly more. In extreme synthesis and simplifying to some extent the interpretation of the results, if random PSO is used rather than DPSO for a number of design variables $N$ smaller or equal to 30, one is suggested to run at least 30 repetitions. For larger $N$, five repetitions may be enough for random PSO to outperform DPSO.

6.2 Hull-form optimization of the DTMB 5415

Figure 4a shows the objective function reduction versus the iteration number $k$, of random PSO and DPSO. For each $k$, the 95% confidence interval, the median and the best solution of random PSO are shown, evaluated using 1,000 optimization runs. DPSO outperforms the median of random PSO, achieving the best solution found by random PSO after 60 iterations ($\Delta f = -5.7\%$). The success probability of random PSO versus DPSO is shown in Fig. 4b. For
the present problem, 20 random PSO repetitions are needed to achieve at least one solution better than DPSO. The probability density function of random PSO results is shown in Fig. 4c, using a kernel density estimate technique. It may be noted how the mode (most probable outcome) is quite far from the DPSO solution, which is close to the random best.

Figure 5a shows the optimal design variables provided by all random PSO runs, their best solution, and DPSO. The solution provided by DPSO is almost coincident with the best solution by random PSO. The corresponding designs are shown in Fig. 5b and compared to the original.

**Figure 4**: Hull-form optimization of the DTMB 5415, algorithm performance

**Figure 5**: Hull-form optimization of the DTMB 5415, optimal design variables
In order to assess the impact of design modifications on the hydrodynamic performances, the solution provided by DPSO has been assessed with RANS and the results were presented in [7] and are shown here in Fig. 6 and Tab. 2. Figure 6 show a significant reduction of the diverging bow wave and a small reduction of the diverging and transverse stern wave. It may be also noted how the shoulder wave is cancelled. Specifically, the optimized shoulder shape induces a high pressure region in correspondence of the first trough of the original hull, causing a phase shift with the reduction of the diverging bow wave and the cancellation of the shoulder wave. The hydrodynamic coefficients for the original and the optimized hulls are finally compared in Tab. 2, confirming that a large part of the resistance reduction stems from the reduction of the piezometric pressure coefficient.

![Figure 6](image)

**Figure 6**: Comparison of wave elevation and pressure distribution on the hull produced by optimized and original configurations at Fr=0.25 [7]

### Table 2: Comparison of hydrodynamic coefficients of optimized and original hulls at Fr=0.25 [7]

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Unit</th>
<th>Original</th>
<th>Optimized</th>
<th>∆%</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_{pp}$</td>
<td></td>
<td>1.38E-03</td>
<td>9.08E-04</td>
<td>-34.0</td>
</tr>
<tr>
<td>$C_h$</td>
<td></td>
<td>0.86E-03</td>
<td>1.24E-03</td>
<td>42.0</td>
</tr>
<tr>
<td>$C_f$</td>
<td></td>
<td>3.16E-03</td>
<td>3.18E-03</td>
<td>0.65</td>
</tr>
<tr>
<td>$C_{mg,x}$</td>
<td></td>
<td>-1.19E-03</td>
<td>-1.35E-03</td>
<td>-13.4</td>
</tr>
<tr>
<td>$C_T$</td>
<td></td>
<td>4.21E-03</td>
<td>3.97E-03</td>
<td>-6.00</td>
</tr>
<tr>
<td>$\sigma/LBP$</td>
<td>deg</td>
<td>-0.11</td>
<td>-0.12</td>
<td>-15.3</td>
</tr>
<tr>
<td>$S_{w,stat}/LBP^2$</td>
<td></td>
<td>1.48E-02</td>
<td>1.50E-02</td>
<td>0.96</td>
</tr>
</tbody>
</table>

7 CONCLUSIONS

In ship hydrodynamics SBDO, the computational cost of the optimization process is strongly affected by the simulation tools (which are usually CPU-time expensive) and by the efficiency of the optimization algorithm. PSO is a widely used global derivative-free optimization algorithm, which has been applied in SBDO for a variety of engineering applications. Its original formulation is stochastic, making use of random coefficients in the particle position update, therefore several optimization runs (repetitions) are needed in order to achieve statistical convergent results. This is often not attainable due to the computational cost of the simulation tools (especially if they are directly used by the optimization algorithm).

For this reason, efficient deterministic PSO (DPSO) formulations have been developed and assessed for SBDO applications, including ship hydrodynamics [8, 9]. The advantage of using deterministic methods is apparent, if one considers that only a single optimization run is needed. But is there any drawback for the effectiveness, efficiency, and robustness of the optimization
procedure? In order to answer this question, a systematic comparison between stochastic and deterministic versions of PSO has been presented and discussed.

The comparison is based on 588 analytical test functions (with dimensionality ranging from two to fifty) and a SBDO problem, aiming the reduction of the total resistance coefficient of the DTMB 5415 in calm water at $Fr=0.25$. More than 500,000 random optimization runs have been performed overall in order to assess the random PSO statistics. The discussion has been based on the probability of success of random PSO versus DPSO.

Analytical test functions have shown that on average DPSO outperforms random PSO with odds of 30/1 for low-dimensional problems (indicatively $N \leq 30$) and 5/1 for high-dimensional problems ($N > 30$). For the ship design SBDO problem ($N = 11$), DPSO outperforms random PSO with odds of 20/1. Moreover, the effectiveness and robustness of DPSO has been further investigated, by comparison of DPSO and best random PSO optima. Design variables provided by DPSO are almost coincident with the best random PSO outcome.

In conclusion, DPSO represents a viable option for SBDO in ship hydrodynamics and its use is advisable whenever computationally expensive analyses are involved in the optimization process. The present conclusion is based on the probability of success of random PSO versus DPSO, considering a relative performance: *is random PSO better than DPSO in solving this problem?* Future work will include extensions of the current analysis, in order to investigate the probability of success of both random PSO and DPSO based on an absolute performance: *how good are random PSO and DPSO in solving this problem?* Absolute performances may represent more effectively engineering requirements addressing simulation tool accuracy, manufacturing tolerance, etc. Finally, the efficiency of the algorithms will be also addressed by analysing the statistics of data and performance profiles [18], with the aim of answering the questions: *how fast are random PSO and DPSO in solving this problem? And how computationally expensive?*

**ACKNOWLEDGEMENTS**

The research is supported by the Office of Naval Research, NICOP grant N62909-15-1-2016, administered by Dr. Woei-Min Lin and, and by the Italian Flagship Project RITMARE, coordinated by the Italian National Research Council and founded by the Italian Ministry of Education.

**REFERENCES**


HULL-FORM OPTIMIZATION OF A 66,000 DWT BULK CARRIER IN IRREGULAR WAVE CONDITION

MARINE 2017

JIN-WON YU*, JUNG-WOO NAM†, INWON LEE† AND JUNG-EUN CHOI*

* Global Core Research Center for Ships and Offshore Plants (GCRC-SOP)
Pusan National University, Busan, South Korea
e-mail: cs.jin@pusan.ac.kr, jechoi@pusan.ac.kr

† Department of Naval Architecture and Ocean Engineering
Pusan National University, Busan, South Korea
Gran Capitán s/n, 08034 Barcelona, Spain
email: jwnam@pusan.ac.kr, inwon@pusan.ac.kr

Key words: Hull-form Optimization, Bulk Carrier, Mean Added Resistance, Parametric Modification Function, PSO,

Abstract. This paper deploys optimization techniques to obtain the optimum hull form of a 66,000 DWT bulk carrier in calm water and in irregular head waves at sea state 6. Parametric modification functions for the bow hull-form variation are SAC shape, section shape (U-V type, DLWL type). Multi-objective functions are applied to minimize the values of wave-making resistance in calm water and mean added resistance in waves. WAVIS version 1.3 is used to obtain wave-making resistance in calm water condition. The modified Fujii and Takahashi’s formula is applied to obtain the added resistance in short waves. The added resistance in long wave is obtained from the potential-flow solver based on the 3-D panel method. And the mean added resistance in irregular head waves is obtained by linear superposition of the wave spectrum and the response function. The PSO (Particle swarm optimization) algorithm is employed for the optimization technique. The resistance and motion characteristics in calm water, in regular head waves and in irregular head waves of the two hull forms are compared. It has been shown that the optimal brings 6.8% reduction in the mean added resistance at sea state 6.

1 INTRODUCTION

Currently shipbuilding companies are asked to develop new hull forms to reduce greenhouse gases. The recent IMO MEPC regulation on EEDI (Energy Efficiency Design Index) makes ship designers to have an interest in the prediction of speed loss due to a real sea condition. Since the added resistance in actual seas is mainly due to winds or waves, it is considered to be effective for the improvement of ship performance in actual seas to reduce the added resistance due to waves ($R_{AW}$). The powering performance of a future ship should be optimized not only for calm water but also in waves. The bow shapes of large and slow speed ships like very large crude carriers (VLCC) or bulk carriers (BC) are generally blunt. A ship with blunt bow can transport more cargo and easier arrangement on the deck than that
with sharp one in equal displacement. This overcomes the demerit of the higher resistance. A ship with blunt bow is usually designed with a focus on lower resistance and higher propulsion efficiency in calm water. Moreover, the reduction of $R_{AW}$ is also to be taken into account at operational condition.

The $R_{AW}$ in short waves is an important factor especially for a large ship’s performance, because the significant frequency of a sea wave spectrum coincides with this range. Guo and Steen (2011) revealed that the fore part of ship has dominant contribution on the $R_{AW}$, that is, the $R_{AW}$ acts on the bow near the free surface dominantly. Many researches showed that the blunt bow shape provides larger $R_{AW}$ (Blok, 1983; Buchner, 1996; Matsumoto, 2002; Hirota et al., 2005; Kuroda et al., 2012; Tvete and Borgen, 2012). A long and protruding bow (named as ‘beak-bow’) reduces the $R_{AW}$, but increases overall length (Matsumoto et al., 2000; Orihara and Miyata, 2003; Hirota et al., 2005). Hirota et al. (2005) showed the results of the favorable effect in waves to use ‘Ax-bow’ and ‘LEADGE-bow’. The Ax-bow, a successor of the beak-bow, is to sharpen the bow only above design load waterline (DLWL). The Ax-bow reduces the wave reflection above the DLWL maintaining the same resistance in calm water (Guo and Steen, 2011; Sadat-Hosseini et al., 2013; Seo et al., 2013). The Ax-bow concept was installed on “Kohyoosan”, a 172,000 DWT Cape size BC (Matsumoto, 2002). The LEADGE-bow is a straightened bow to fill up the gap between the Ax-bow and the bulb. The whole bow line including under the DLWL is sharpened. Due to this the bow was expected to reduce the added resistance in both ballast and full load conditions. Hwang et al. (2013) applied the design concepts of Ax- and LEADGE-bow to 300k DWT VLCC (KVLCC2).

SEA-Arrow (Sharp Entrance Angle bow as an Arrow) is developed and applied to medium-speed ships such as LPG carriers (Ebira et al., 2004). However, in the case of a ship with relatively sharp bow, such as high speed fine ship, the Ax-bow does not reduce the $R_{AW}$. In such ships, bow flare angle is a useful design parameter. The $R_{AW}$ increases with the bow flare angle (Fang, 1995; Orihara and Miyata, 2003; Kihara et al., 2005; Fang et al., 2013; Jeong et al., 2013). If the vessels encounter short waves most of the time, a sharper bow may be optimal. However, if the encountered waves are in the radiation regime the majority of the operating time a sharper bow is expected to be less, as the motion characteristics are most important in this range. The X-bow of backward sloping bow is developed for not only reducing the $R_{AW}$ but also improving motion characteristics of offshore vessels (Ulstein Group, 2005). The STX bow consists of three parts, i.e., A, B and C (Tvete and Borgen, 2012). The upper bow portion, C, is stretched forward making it sharper. This makes it possible to reduce the flare angles. The middle part, B, comprises a blunt shaped surface of transition area. And the lower part, A, is kept more or less as conventional hulls to minimize the calm water resistance.

The hull-form optimization to satisfy the objective functions taking the wave effect into account through the simulation-based design (SBD) has not been widely applied. Most of the objective functions are related to the seakeeping performances; Wigley and Series 60 with minimum bow vertical motion (Bagheri et al., 2014), SR175 container ship with minimum heave and pitch motions (Campana et al., 2009), ferry with minimum wave height in calm water and absolute vertical acceleration (Grigoropoulos and Chalkias, 2010), combatant ship DTM 5415 with total resistance and seakeeping (Tahara et al., 2008; Kim et al., 2010).

In this paper, the hull-form optimization in calm water and in short wave through the SBD is proposed. The objective ship is a 66k DWT BC. Hull forms are varied by parametric
modification functions. Two objective functions are taken into account; minimum wave-making resistance in calm water and mean added resistance ($R_{AW}$) in short crested irregular head waves. The varied hull forms are coupled with the deterministic particle swarm optimization.

2 OBJECTIVE SHIP

The objective ship is 66k DWT BC. Two hull forms had been developed. The former, designed by DSME (Daewoo Shipbuilding & Marine Engineering Co., Ltd.) and KAIST (Korea Advanced Institute of Science and Technology), is initial hull form, which bow hull form is applied to the concept of LEADGE bow to reduce the $R_{AW}$. The body plan and side view of the initial hull forms are presented in Fig. 1. The principal dimensions at the full-load draft are listed in Table 1. The design speed ($V_S$) at the full-load draft is 14.5 knots and the Froude number ($F_N$)=0.170. The $F_N$ is non-dimensionalized by the $V_S$ and LPP.

![Figure 1: Body plan and side view of the initial hull form](image)

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>LOA</td>
<td>200.0</td>
<td></td>
</tr>
<tr>
<td>LPP</td>
<td>196.0</td>
<td></td>
</tr>
<tr>
<td>LWL</td>
<td>200.0</td>
<td></td>
</tr>
<tr>
<td>B</td>
<td>36.0</td>
<td></td>
</tr>
<tr>
<td>T</td>
<td>11.2</td>
<td></td>
</tr>
<tr>
<td>WSA</td>
<td>9,773</td>
<td></td>
</tr>
<tr>
<td>∇</td>
<td>64,472</td>
<td></td>
</tr>
<tr>
<td>LCB</td>
<td>5.27</td>
<td></td>
</tr>
<tr>
<td>KG</td>
<td>7.02</td>
<td></td>
</tr>
<tr>
<td>WPA</td>
<td>6,501</td>
<td></td>
</tr>
</tbody>
</table>

3 PROBLEM FORMULATION

The mathematical formulation of the optimization problem is expressed as

$$\text{Minimize } [f_1(\vec{x}), f_2(\vec{x}), \ldots, f_k(\vec{x})]$$

Subject to the equality and inequality constraints

$$h_j(\vec{x}) = 0, \quad j = 1, \ldots, p$$

$$g_j(\vec{x}) < 0, \quad j = 1, \ldots, q$$
where \( f_i(\vec{x}) \) is the objective function, \( K \) is the number of objective functions, \( p \) is the number of equality constraints, \( q \) is the number of inequality constraints and \( \vec{x} = (x_1, x_2, \ldots, x_N) \subseteq S \) is a solution or design variable. The search space \( S \) is defined as an \( N \)-dimensional rectangle in \( \mathbb{R}^N \) (domains of variables defined by their lower and upper bounds):

\[
x_{i}^l \leq x_i \leq x_{i}^u, \quad i = 1, \ldots, N
\] (4) 

The constraints define the feasible area. This means that if the design variables vector \( \vec{x} \) be in agreement with all constraints \( h_i(\vec{x}) \) (equality constraint) and \( g_j(\vec{x}) \) (inequality constraint), it belongs to the feasible area.

In this study, the bow hull form is to be optimized, whereas the stern hull form remains fixed. The objective functions are to minimize wave-making resistance in calm water (\( R_W \)) and the \( AWR \) in sea state (SS) 6 at \( V_M=1.292 \text{m/s} \). The constraints are principal particulars of \( \text{LOA}, B \) and \( T \). The displacement is an inequality constraint, which is kept within \( \pm 1\% \) of the initial value.

4 ESTIMATION OBJECTIVE FUNCTIONS AND HULL FORM VARIATION

The objective functions are to minimize the \( R_W \) in calm water and the \( \overline{R_{AW}} \) in short crested irregular head waves.

The \( R_W \) may be obtained from the potential-flow solver which is utilized WAVIS v.1.3. The details and formulations of the numerical methodologies are well described in the works of Kim et al. (1998, 2000). In the present work, 1923 panels on the hull and 1815 panels on the free surface are used. This has been deemed appropriate to identify the proper trends of the objective functions. During the computation the ship is fixed to sink and trim, and the nonlinear free-surface boundary condition is applied.

The \( R_{AW} \) in short wave is primarily due to wave reflection. In the short wave range, the numerical methods based on the potential-flow theory may not be reliable since the \( R_{AW} \) due to ship motion is almost negligible and the \( R_{AW} \) due to wave reflection (\( R_{AW \text{ref.}} \)) is dominant. Semi-empirical formula (Kuroda et al., 2008) is used in this work for calculation of \( R_{AW} \) in regular short wave.

The \( R_{AW} \) in regular long waves is mainly due to ship motion. The 3-D panel source distribution method based on the frequency-domain approach is used. The \( R_{AW} \) is obtained from the direct pressure integration. The details and the formulations of the numerical methodologies are extensively documented in Chun (1992).

The \( \overline{R_{AW}} \) in short crested irregular head waves is calculated by linear superposition of the wave spectrum \( S(\omega) \) and the response function of the \( R_{AW} \) (Strom-Tejsen et al., 1973).

\[
\overline{R_{AW}} = 2 \int_0^\infty \frac{R_{AW}(\omega)}{\sqrt{2}} S(\omega) d\omega
\] (5)

\( \overline{R_{AW}} \) in short crested irregular head waves is calculated by linear superposition of the wave spectrum \( S(\omega) \) and the response function of the \( R_{AW} \) (Strom-Tejsen et al., 1973).

\[
S(\omega) = \frac{A}{\omega^5} \exp\left(-\frac{B}{\omega^2}\right) \text{[m}^2\text{s]}
\] (6)
where $\zeta$ is wave amplitude, $A = 173H_{1/3}^2 / T_1^4$, $H_{1/3}$ is significant wave height, $B = 691/T_1^4$, and $T_1$ is the averaged period.

A designer-friendly parametric modification tool is adopted for modifying the hull form according to the classical naval architect’s approach as well as the office design practice.

The parametric modification function is superimposed on the original hull ($H_{\text{old}}$) to obtain modified geometry ($H_{\text{new}}$):

$$H_{\text{new}}(X,Y,Z) = H_{\text{old}}(X,Y,Z) + r^{(f)}(X) \cdot s^{(m)}(Y) \cdot t^{(n)}(Z) \quad (7)$$

where $r^{(f)}(X)$, $s^{(m)}(Y)$ and $t^{(n)}(Z)$ are the parametric modification functions defined as polynomials along the X, Y and Z directions, respectively. The superscripts $(f)$, $(m)$ and $(n)$ are the orders of polynomials. Here, a local coordinate (X, Y, Z) is applied, where the positive X direction goes from the AP to the FP, and the positive Z direction is vertical from the hull bottom. The modified geometry is obtained using the perturbation with specific direction depending on the design parameters. Sectional area curve (SAC) and section shape of DLWL type are used as modification functions.

The SAC and section shape of DLWL parametric modification functions are:

$$X_{\text{new}} = X_{\text{old}} + r^{(6)}(X) \quad (8)$$

$$Y_{\text{new}} = Y_{\text{old}} + r^{(4)}(X) \cdot s^{(5)}(Y) \cdot t^{(1/3)(1/2)}(Z) \quad (9)$$

Details are well documented in Park et al. (2015) and Kim et al. (2016).

5 HULL FORM OPTIMIZATION

The particle swarm optimization (PSO) algorithm is applied for the optimization technique, which is a gradient-free global optimization algorithm. The PSO assumed that each individual in the particles swarm is composed of three N-dimensional vectors, where N is the dimensionality of the search space. These are the current position ($\vec{x}_i$), previous best position ($\vec{p}_i$), and velocity ($\vec{v}_i$). A particle swarm is composed of $N_v$ number of particles, the position of the number i particle is expressed as $\vec{x}_i = [x_{i1}, x_{i2}, \ldots, x_{iN}]$ and so the velocity is $\vec{v}_i = [v_{i1}, v_{i2}, \ldots, v_{iN}]$. The best position find by the number i particle is $\vec{p}_i = [p_{i1}, p_{i2}, \ldots, p_{iN}]$ and the best position find by the whole particles is expressed as $\vec{p}_g = [p_{g1}, p_{g2}, \ldots, p_{gN}]$. The basic algorithm is simple as follows:

- Step 0 (Initialize): Distribute a set of particles inside the design space. Evaluate the objective function in the particles’ position and find the best location ($p_b$). Note that the effective number and distribution of the initial particles significantly affect the results in the PSO algorithm.

- Step 1 (Compute particle’s velocity): At the step $k+1$, calculate the velocity vector $\vec{v}_i$ for each particle i using the equation:

$$\vec{v}_i^{k+1} = \chi \left[ w \vec{v}_i^k + c_1 \vec{p}_i^k - x_i^k \right] + c_2 \vec{p}_g^k - x_i^k \quad (10)$$

where $\chi$ is a speed limit and w is the inertia of the particles controlling the impact of the previous velocities onto the current one. The second and third terms, with weights $c_1$ and $c_2$,
are the individual and collective contributions, respectively and finally; \( r_1 \) and \( r_2 \) are random coefficients uniformly distributed in \([0,1]\).

- Step 2 (Update position): Update the position of each particle
  \[
  x_i^{k+1} = x_i^k + v_i^k
  \]
  \quad (11)

- Step 3 (Check convergence): Go to Step 1 and repeat until some convergence criterion (e.g. the maximum distance among the particles, a condition on the velocity) comes to a match.

Experimental results indicate that a large value of the inertia \( w \) promotes a wide exploration of the global search space. Hence \( w \) is initially set to a high value and then gradually decreased (\( w^{k+1} = K \cdot w^k \), with \( K < 1 \)) to facilitate the fine-tuning of the current search area. The details and formulations of the numerical methodologies are well described in the works of Kim et al. (2016). The set of parameters adopted in the computations are listed in Table 2.

**Table 2: Particle swarm optimization parameters**

<table>
<thead>
<tr>
<th>PSO parameters</th>
<th>Present problem</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constriction parameter (speed limit)</td>
<td>( X )</td>
</tr>
<tr>
<td>Initial inertia weight</td>
<td>( w_0 )</td>
</tr>
<tr>
<td>Decreasing coefficient for the inertia</td>
<td>( K )</td>
</tr>
<tr>
<td>Individual parameter</td>
<td>( c_1 )</td>
</tr>
<tr>
<td>Social parameter</td>
<td>( c_2 )</td>
</tr>
</tbody>
</table>

Fig. 2 shows the distributions of all the swarm particles and the Pareto optimal set. Distribution of the particles is concentrated around a certain point. Among swarm particles, optimal solution is chosen, that sum of the \( R_W \) and \( AWR \) decrease is maximum. The optimal hull form (hereafter ‘the optimal’) is obtained at \( \Delta X_{\text{max}} = -0.248 \) and \( \Delta Y_{\text{max}} = -0.0698 \) with \( \Delta R_W = 0.003N \) and \( \Delta \overline{AWR} = 1.505N \).

The \( R_W \) and \( \overline{AWR} \) in SS 6, and the displacement and wetted surface at model scale of the initial and the optimal are compared in Table 3. RR\% is a percentage reduction ratio of the value of the optimal to that of the initial. Decreasing amount of the \( R_W \) is small. However, the decreasing amount of the \( \overline{AWR} \) is much greater than that of the \( R_W \). Displacement of the optimal is reduced by 0.8%, which is within the inequality constraint \( \pm 1\% \) of the original value. The WSA also shows the same tendency.

The body plans, the shape of DLWL and three-dimensional view at bow region of the initial and the optimal are displayed in Fig. 3. The cross-sectional shape is changed to U type and section shape of DWL is varied sharply in the bow region, forward of station 17. But remains unchanged aft of station 17.

The wave patterns in calm water around the initial and the optimal are displayed in Fig. 4. The divergent wave is clearly shown. The wave elevations near the fore-shoulder part of the optimal are a little lower than those of the initial. This is due to slender waterline at fore-shoulder part near design draft.
Figure 2: Swarm particles from multi objective optimization of initial hull

Table 3: Wave-making resistance in calm water, mean added resistance in SS 6, displacement and wetted surface at model scale

<table>
<thead>
<tr>
<th>Hull form</th>
<th>Initial</th>
<th>Optimal</th>
<th>Diff.</th>
<th>RR%</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R_w$ [N]</td>
<td>1.734</td>
<td>1.731</td>
<td>-0.003</td>
<td>-0.2</td>
</tr>
<tr>
<td>$R_{AW}$ [N]</td>
<td>22.110</td>
<td>20.605</td>
<td>-1.505</td>
<td>-6.8</td>
</tr>
<tr>
<td>$\nabla$ [m$^3$]</td>
<td>1.750</td>
<td>1.736</td>
<td>-0.014</td>
<td>0.8</td>
</tr>
<tr>
<td>WSA [m$^2$]</td>
<td>8.935</td>
<td>8.886</td>
<td>-0.039</td>
<td>0.5</td>
</tr>
</tbody>
</table>

Figure 3: Swarm particles from multi objective optimization of initial hull
Figure 4: Comparison of wave patterns in calm water at design speed

Figure 5: Added resistance coefficients and motion RAOs in head sea
Fig. 5 displays the RAOs of the non-dimensional added resistance, heave and pitch motions, where $\xi_3$ and $\xi_5$ are heave and pitch amplitudes, and $k$ is wave number. ITTC wave spectra are appended at Fig. 6(a). In short $\lambda$ ($\lambda/LPP<0.8$), the non-dimensional $R_{AW}$ is nearly constant. The the non-dimensional $R_{AW}$ increases as $\lambda$ increases before the peak value. After the peak value, the the non-dimensional $R_{AW}$ decreases as $\lambda$ increases. The peak value occurs around $\lambda/LPP=1.0$~1.2. There is little difference in the heave and pitch amplitude.

The $R_W$, $R_{AW}$ at $\lambda/LPP=0.5$ and $R_{AW}$ at SS 4-6 of the initial and the optimal are compared in Table 4. The optimal is greatly enhanced in the resistance performance in waves. However, the $R_W$ of the optimal is reduced by 0.2%. The $R_{AW}$ is also reduced by 15.1%. The $R_{AW}$ at SS 6 of the optimal is reduced by 6.8%; and the SS 4 and 5 are similar showing the RR%≈4~8%.

Table 2: Particle swarm optimization parameters

<table>
<thead>
<tr>
<th>Wave condition</th>
<th>$R_W$, $R_{AW}$, $R_{AW}$ [N]</th>
<th>Wave condition</th>
<th>$R_W$, $R_{AW}$, $R_{AW}$ [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>H$_{1/3}$ [m]</td>
<td>T$_1$ [sec]</td>
<td>Initial Value</td>
<td>Optimal Value</td>
</tr>
<tr>
<td>Calm water ($R_W$)</td>
<td>-</td>
<td>1.734</td>
<td>1.731</td>
</tr>
<tr>
<td>Regular wave ($R_{AW}$)</td>
<td>$\lambda/L=0.5$</td>
<td>5.865</td>
<td>4.982</td>
</tr>
<tr>
<td>Irregular wave ($R_{AW}$)</td>
<td>SS 4: 0.056, 0.916</td>
<td>0.705</td>
<td>0.676</td>
</tr>
<tr>
<td></td>
<td>SS 5: 0.098, 1.206</td>
<td>6.047</td>
<td>5.544</td>
</tr>
<tr>
<td></td>
<td>SS 6: 0.150, 1.495</td>
<td>22.110</td>
<td>20.605</td>
</tr>
</tbody>
</table>

6 CONCLUSIONS

- A practical hull-form optimization technique to minimize the values of wave-making resistance in calm water and mean added resistance in short crested irregular head waves at sea state 6 has been introduced. The hull form including above design load waterline is readily varied using the parametric modification functions for the SAC and the section shape of DLWL. The Pareto optimal set has been obtained using the deterministic optimization technique of PSO.

- The optimal of a 66,000 DWT bulk carrier features more slender at fore-shoulder part. The optimal brings 0.2% reduction in the wave-making resistance and 6.8% reduction in the mean added resistance at sea state 6 in comparison with those of the initial hull form. There is little difference in the heave and pitch amplitude.

- Designer friendly hull-form variation and optimization techniques by taking resistance performances of not only in calm water but also in waves into account at shipyard are developed. Hull form designer will easily acquire objective information and save hull-form development period.

ACKNOWLEDGEMENTS

This work has been supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) through GCRC-SOP (No. 2011-0030013), to which deep gratitude is expressed.
REFERENCES


HULL-FORM OPTIMIZATION OF A LUXURY YACHT UNDER DETERMINISTIC AND STOCHASTIC OPERATING CONDITIONS VIA GLOBAL DERIVATIVE-FREE ALGORITHMS

CECILIA LEOTARDI

National Research Council-Marine Technology Research Institute (CNR-INSEAN)
Via di Vallerano 139, 00128 Rome, Italy
e-mail: cecilia.leotardi@cnr.it

Key words: Global derivative-free optimization, deterministic optimization, robust optimization, simulation-based design, ship hydrodynamics

Abstract. Simulation-based design optimization (SBDO) techniques are used in the shape design of complex engineering systems. SBDO methods integrate an optimization algorithm, a tool for the design modification, and analysis tools. In the context of ship/ocean applications the objective function is often noisy, its derivatives are not directly provided and local minima cannot be excluded, therefore global derivative-free algorithms are widely used. The objective of this work is to investigate the efficiency of three global deterministic derivative-free optimization algorithms for the deterministic and stochastic hull-form optimization of a luxury yacht. Particle Swarm Optimization (DPSO), Dolphin Pod Optimization (DPO), and DIviding RECTangles (DIRECT) are applied to reduce the total resistance over a variety of conditions. The approach includes a comparison of the performances of the optimization algorithms, based on deterministic results for two separate operating conditions. DPSO is identified as the most promising optimization algorithm and is used for the robust design optimization (RDO) performed considering a stochastic variation of the cruise speed with uniform distribution within a speed range from 8 to 16 kn. The resistance curve of deterministic and robust solutions is finally presented.

1 INTRODUCTION

Simulation-based design analysis and optimization (SBDO) techniques represent an actual paradigm to assist designers in assessing and designing complex engineering systems, where both objectives and constraints often involve several concurrent disciplines. SBDO for shape design has been widely used for diverse engineering applications, such as aerospace, automotive, and naval, where the vehicle performances (e.g., aerodynamic and aeroelasticity [1, 2], hydrodynamics and hydro-structural interaction [3, 4, 5], structures [6], etc.) strongly depend on the shape design. In this context, automatic SBDO approaches need to efficiently integrate three elements:
an optimization algorithm, a geometry modification and automatic meshing tools, and simulation solvers.

Real life applications are affected by several sources of uncertainty, such as environmental and/or operating conditions. Specifically, uncertainty quantification (UQ) methods are needed to evaluate the effects of stochastic input parameters on the relevant outputs, which can be represented in terms of expected value, standard deviation, and probability distribution [7, 8, 9]. Integrating UQ within the design optimization process leads to robust design optimization (RDO) formulations. SBD/RDO still represents a significant theoretical, algorithmic, and computational challenge, especially when high-fidelity and/or multi-disciplinary solvers are employed [10, 11].

The objective of the present work is to investigate the efficiency of three global deterministic derivative-free optimization algorithms in the hull-form optimization of a luxury yacht under deterministic and stochastic operating conditions. The interest in investigating and using deterministic global derivative-free algorithms for deterministic and/or robust simulation-based shape design optimization for ship/ocean applications depends on several concurrent issues. Objective and constraints are often solved using black-box tools through systems of partial differential equations, they might be noisy and derivatives are not directly provided. Furthermore, the presence of local minima in the search space cannot be excluded a priori.

Particle Swarm Optimization (DPSO) [12], Dolphin Pod Optimization (DPO) [13], and Di-viding RECTangles (DIRECT) [14] are applied to reduce the total resistance of a 35m length overall yacht. The hull form is modified using two design variables, controlling global modification functions. Design constraints include fixed displacement, fixed length between perpendiculars, fixed beam and limited (5%) variation of the draft. Two deterministic operating conditions (namely, cruise speed equal to 10 and 14 kn) are considered separately, along with a stochastic variation of the cruise speed (uniformly distributed from 8 to 16 kn) for the RDO. A stationary 2DOF problem with free surface is studied by potential flow simulations [15]. Specifically, the ship advances in calm water and is free to heave and pitch. A quasi Monte Carlo method coupled with a deterministic variant of the Latin Hypercube Sampling (LHS, [10, 11]) is used for UQ within RDO.

For the sake of confidentiality, the original geometry of the yacht hull and its optimized versions are not shown in the current paper.

2 DETERMINISTIC VERSUS ROBUST OPTIMIZATION: PROBLEM FORMULATION

The general deterministic single-objective design optimization problem may be formulated, within the SBD context, as

\[
\text{maximize or minimize } f(\mathbf{x}), \quad \mathbf{x} \in X \subseteq \mathbb{R}^{N_{DV}}
\]  

where \( f \), provided by simulations, is the deterministic objective function and the vector \( \mathbf{x} \) collects the \( N_{DV} \) deterministic design variables. Equality and inequality functional constraints may be applied to the problem formulated in Eq. 1, if required. Such a formulation implies that all the inputs of the simulation tools are deterministic and the design optimization is performed for specific design, operating, and environmental conditions.
When dealing with real life applications, the problem presented in Eq. 1 is affected by several sources of uncertainties, such as those related to operating and/or environmental conditions, therefore UQ methods need to be embedded within the optimization process. In RDO the effects of the input uncertainties may be evaluated using diverse formulations of the original objective function. The expected value, the standard deviation, the cumulative distribution function or the weighted sum of expected value and standard deviation of the deterministic objective function might be used to assess the RDO problem in its general form. The current work focuses on the expected value only, correspondingly the problem in Eq. 1 is reformulated as

$$\text{maximize or minimize } \mu(f) = \int_Y f(x, y) p(y) dy, \quad x \in X \subseteq \mathbb{R}^{NDV}$$

where $\mu$ is the expected value of the original deterministic objective function $f$, $x$ collects the $NDV$ deterministic design variables, and $p(y)$ is the probability density function of the stochastic operating and environmental conditions, collected in $y \in Y$. Functional constraints may be applied and included in the formulation, if required.

3 SBD OPTIMIZATION PROBLEM AND TOOLBOX AT A GLANCE

The objective function for the problem in Eq. 1 is the total resistance of the yacht in calm water at 10 and 14 kn, whereas the objective function for the problem in Eq. 2 is the expected value of the total resistance in calm water, evaluated over a stochastic speed $y$, with $y \in [8; 16]$ kn following a uniform probability density function.

The SBD optimization toolbox includes three essential and interconnected elements: (a) the optimization algorithm, (b) a tool for the design modification, and (c) the analysis tools. For the current application three deterministic, single-objective, derivative-free, and global optimization algorithms are used, the shape modifications are performed using orthogonal basis functions, the hydrodynamic performances are evaluated using a potential flow solver, and for the RDO problem, the UQ is performed using a quasi MC method coupled with deterministic LHS. An overview of the tools implemented is presented in the following.

3.1 Global Derivative-Free Optimization Algorithms

The numerical optimization is performed using DPSO algorithm, DPO, and DIRECT, which are briefly recalled in the following.

3.1.1 Deterministic Particle Swarm Optimization [DPSO]

PSO, originally introduced by [16], is based on the social-behavior metaphor of a flock of birds or a swarm of bees searching for food. PSO belongs to the class of heuristic algorithms for single-objective evolutionary derivative-free global optimization. In order to make PSO more efficient for SBDO, a deterministic version of the algorithm, namely DPSO, was formulated by [17] and further investigated in [12]. Accordingly, the location $x^i_j$ of the $j$-th particle at the $i$-th iteration is evaluated as

$$\begin{align*}
    v^{i+1}_j &= \chi[v^i_j + c_1(x_{gb}^i - x^i_j) + c_2(x^i_j - x^i_j)] \\
    x^{i+1}_j &= x^i_j + v^{i+1}_j
\end{align*}$$

(3)
where \( x_{j,pb} \) is the personal optimum or cognitive term (the best position ever visited by the \( j \)-th particle), \( x_{gb} \) is the global or social optimum (the overall best position ever visited by all the particles), \( \chi \) is the constriction factor and \( c_1 \) and \( c_2 \) are respectively the cognitive and the social learning rate coefficients.

DPSO performances depend on the number of swarm particles interacting during the optimization, on the initialization of the particles in terms of position and speed in the search space, and on the set of coefficients defining the personal or global behaviour of the swarm dynamics. Following [12], the swarm dimension is set to \( 4 \cdot N_{DV} \), the swarm is initialized using Hammersley sequence sampling (HSS) [18] over the design variables domain and its boundary with non-null velocity, and the coefficients are defined as \( \chi = 0.721, c_1 = c_2 = 1.655 \) [19]. For the current application the number of function evaluations is set to \( N_{feval} = 128 \cdot N_{DV} \).

### 3.1.2 Dolphin Pod Optimization [DPO]

DPO algorithm was introduced by [13], based on the metaphor of a pod of dolphins. The \( j \)-th dolphin position \( x^j_i \) and velocity vector \( v^j_i \) at the \( i \)-th iteration are respectively defined as

\[
\begin{align*}
    v^{j+1}_i &= (1 - \xi \Delta t)v^j_i + \Delta t(-k \delta^j + h \varphi^j) \\
    x^{j+1}_i &= x^n_i + v^{j+1}_i \Delta t
\end{align*}
\]

The coefficient of Eq. 4 are defined as

\[
k = h = \frac{q}{N_d}, \quad \Delta t = \frac{\Delta t_{\text{max}}}{p}, \quad \xi \Delta t < 1
\]

where \( N_d \) is the number of dolphins, \( q \) defines the weights for the attraction forces \( \delta^j \) (congregation) and \( \varphi^j \) (self-awareness, communication and memory), whereas \( p \) defines the integration time step \( \Delta t \).

DPO performances depend on the number of dolphins interacting during the optimization, on its initialization in terms of position and speed in the design space, and on the set of coefficients that control its dynamic. Following the guideline suggested in [13] the pod dimension is set to \( N_d = 4 \cdot N_{DV} \), the pod is initialised using HSS on the domain, and the coefficients are defined as \( \xi = 1.00, q = 1.00, p = 8.00, \alpha = 0.50 \). For the current application the number of function evaluations is set to \( N_{feval} = 128 \cdot N_{DV} \).

### 3.1.3 Dividing RECTangles [DIRECT]

DIRECT is a sampling deterministic global derivative-free optimization algorithm and a modification of the Lipschitzian optimization method [14, 20, 21]. The optimization process starts transforming the domain \( D \) of the problem into the unit hyper-cube \( D \). At the first step \( f(x) \) is evaluated at the center \( (c) \) of the search domain; the hyper-cube is then partitioned into a set of smaller hyper-rectangles and \( f(x) \) is evaluated at their centers. The partition of \( D \) at iteration \( k \) is defined as

\[
\mathcal{H}_k = \{ D_i : i \in I_k \}, \quad \text{with} \quad D_i = \{ x \in \mathbb{R}^{N_{DV}} : l_j^{(i)} \leq x_j \leq u_j^{(i)} , j = 1, \ldots, N_{DV}, \forall i \in I_k \}
\]
where $\xi^0_j$ and $\eta^0_j \in [0,1]$, with $i \in I_k$, are the lower and upper bounds defining the hyper-rectangle $D_i$ and $I_k$ is the set of indices identifying the subsets defining the current partition. At a generic $k$-th iteration, starting from the current partition $H_k$ of $D$, a new partition, $H_{k+1}$, is built subdividing a set of promising hyper-rectangles of the previous one. The identification of potentially optimal hyper-rectangles is based on the measure of the hyper-rectangle itself and on the value of $f(x)$ at its center $c_i$. The refinement of the partition continues until a prescribed number of function evaluations is performed, or another stopping criterion is satisfied. The minimum of $f(x)$ over all the centers of the final partition, and the corresponding centers, provide an approximate solution to the problem. For the current application the number of function evaluations is again set to $N_{feval} = 128 \cdot N_{DV}$.

3.2 Shape Modification Method

Shape modifications are represented in terms of orthogonal basis functions $\psi_j (j = 1, ..., N_{DV})$, defined over surface-body patches as [20, 21, 22]

$$\psi_j (\xi, \eta) := \alpha_j \sin \left( \frac{p_j \pi \xi}{A_j} + \phi_j \right) \sin \left( \frac{q_j \pi \eta}{B_j} + \chi_j \right) e_{k(j)} \quad (\xi, \eta) \in [0; A] \times [0; B]$$

(7)

where $\alpha_j$ is the $j$-th (dimensional) design variable; $p_j$ and $q_j$ define the order of the basis function in $\xi$ and $\eta$ direction respectively; $\phi_j$ and $\chi_j$ are the corresponding spatial phases; $A_j$ and $B_j$ are the patch extension in $\xi$ and $\eta$ respectively, and $e_{k(j)}$ is a unit vector. Modifications may be applied in $x$, $y$ or $z$ direction ($k(j) = 1, 2, 3$ respectively).

In this work two (normalized) design variables, $x_1 = \alpha_1/2$ and $x_2 = \alpha_2$, are used. The shape modifications are obtained as per Eq. 7 with $j = 1, 2$ and $k = 3$. The associated parameters are

$p_1 = 2.0, \quad \phi_1 = 0.0, \quad q_1 = 1.0, \quad \chi_1 = 0.0, \quad \alpha_1 \in [-2.0; 2.0] \quad \text{and} \quad x_1 \in [-1.0; 1.0]; \quad p_2 = 1.0, \quad \phi_2 = 0.0, \quad q_2 = 2.0, \quad \chi_2 = 0.0, \quad \alpha_2 \in [-1.0; 1.0] \quad \text{and} \quad x_2 \in [-1.0; 1.0]$. Geometric constraints include fixed length between perpendiculars ($L_{BP}$), fixed displacement, and fixed beam, whereas a 5% change in draft is allowed.

3.3 Hydrodynamic analysis

The hydrodynamics is solved using the code WARP (WAve Resistance Program), developed at CNR-INSEAN. For details of equations, numerical implementation and validation the reader is addressed to [15]. Specifically, the wave resistance computations are based on the double-model linear potential flow (PF) theory [23], whereas the frictional resistance is estimated using a flat-plate approximation, based on the local Reynolds number [24].

The following assumptions are used for the numerical investigations. The numerical model, free to sink and trim (2DOFs problem), advances in calm water with $v \in [8; 16]$ kn. Its reference length $L_{BP}$ equals to 34.5 m ($0.224 \leq Fr \leq 0.447$) and the displacement $\Delta$ equals to 240 tons. The fluid conditions are $\rho = 1000$ kg/m$^3$, $\nu = 1.1E-06$ m$^2$/s and $g = 9.81$ m/s$^2$.

PF calculations are performed using a 150x50 panel grid for the hull surface. The computational domain for the free surface is defined within 1 hull length upstream, 3 lengths downstream and 1.5 lengths aside, discretized with 30x44, 90x44 and 30x44 surface panels, respectively. Grid convergence analysis is provided in [25].


3.4 UQ method

The integral in Eq. 2 is approximated using a quasi MC method as

\[
\mu(f) = \frac{1}{N_{UQ}} \sum_{i=1}^{N_{UQ}} f(x, y_i)
\]  

(8)

A deterministic variant of the LHS method is applied, dividing the uncertain parameter domain (1D) with \(N_{UQ} = 2^h + 1\) \((h \in \mathbb{N}^+)\) evenly spaced items \([10, 11]\).

4 NUMERICAL RESULTS

Numerical results are organized as follows. Subsection 4.1 shows the results of the deterministic design optimization aimed at minimizing the total resistance, \(R_T\), in two operating conditions \(i.e. v=10\) and \(v=14\) kn, respectively indicated as Problem I and II in the following). Moreover the most promising optimization algorithm is identified, based on deterministic design optimization results. The RDO results are shown in subsection 4.2 and include a parametric performance analysis of the RDO solution and the best performing deterministic configurations.

4.1 Deterministic optimization

For Problem I, DPSO, DPO, and DIRECT have a similar objective function reduction (differences are below 0.5%) even if DPSO and DPO show a faster convergence to the global minimum, as depicted in Fig. 1a. The final configurations found by the three algorithms are very close in terms of design variables, as shown in Fig. 2a. The deterministic optimization results, summarized in Tab. 1, show that the three algorithms achieved approximately the same total resistance reduction equal about to 11.60%. Figure 3a shows a parametric performance analysis on the three optimized configurations compared to the original one, versus the cruise speed. As desired, the optimized shapes outperform the original for the design speed \((v = 10\) kn). For higher speeds, the optimized hulls show a total resistance larger than the original.

![Figure 1: Deterministic optimization, algorithm convergence](image-url)
Figure 2: Deterministic optimization, optimal design variables

(a) Problem I optimized hull (v = 10 kn)  (b) Problem II optimized hull (v = 14 kn)

Figure 3: Resistance curve of optimal configurations

Table 1: Deterministic Problem I results (v = 10 kn)

<table>
<thead>
<tr>
<th>ID</th>
<th>$x_1$</th>
<th>$x_2$</th>
<th>$\Delta R_T$%</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>0.000</td>
<td>0.000</td>
<td>-</td>
</tr>
<tr>
<td>DPSO</td>
<td>-1.000</td>
<td>-0.159</td>
<td>-11.65</td>
</tr>
<tr>
<td>DPO</td>
<td>-1.000</td>
<td>-0.167</td>
<td>-11.60</td>
</tr>
<tr>
<td>DIRECT</td>
<td>-1.000</td>
<td>-0.162</td>
<td>-11.62</td>
</tr>
</tbody>
</table>

For Problem II, DPSO, DPO, and DIRECT achieve a comparable objective function reduction (the differences are smaller than 1%) even if DPSO shows a faster convergence to the global minimum and achieves a better solution than DPO and DIRECT, as depicted in Fig. 1b. The final configurations found by DPSO and DIRECT are close in terms of design variables (see Fig. 2b), whereas DPO design variables values are slightly different. The deterministic optimization
results are summarized in Tab. 2, showing a total resistance reduction of 5.45%, achieved by DPSO. Figure 3b shows a parametric performance analysis of the three optimized configurations compared to the original one, versus the cruise speed. As desired, the three optimized shapes outperform the original for the design speed (v = 14 kn). For lower speeds, their total resistance is found larger than the original.

<table>
<thead>
<tr>
<th>ID</th>
<th>x₁</th>
<th>x₂</th>
<th>∆Rₜ (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>0.000</td>
<td>0.000</td>
<td>-</td>
</tr>
<tr>
<td>DPSO</td>
<td>0.911</td>
<td>-0.503</td>
<td>-5.45</td>
</tr>
<tr>
<td>DPO</td>
<td>0.806</td>
<td>-0.308</td>
<td>-4.66</td>
</tr>
<tr>
<td>DIRECT</td>
<td>0.850</td>
<td>-0.404</td>
<td>-4.98</td>
</tr>
</tbody>
</table>

Finally, Fig. 4 shows the comparison between the wave elevation of the original and the DPSO optimized configurations for Problem I (v = 10 kn) and II (v = 14 kn), respectively. It may be noted how, for Problem I, the optimized hull reduces bow and stern diverging Kelvin waves and stern transverse wave. For Problem II, the optimized hull only reduces the bow wave while slightly increases stern diverging and transverse waves, resulting in a smaller performance improvement than Problem I.

4.2 Robust optimization

The UQ convergence is studied versus the number of samples $N_{UQ} = 2^h + 1$, $h \in \mathbb{N}$, for the design variables at centre domain value ($x₁ = x₂ = 0$). Figure 5a shows the solution change of the expected value of the total resistance ($\Delta \mu/\mu$) versus $N_{UQ}$. A benchmark tolerance for solution changes is set to $10^{-3}$, correspondingly a number of UQ samples $N_{UQ} = 33$ is identified.

Figure 5b represents the DPSO convergence history. DPSO achieves an objective function reduction of 7.6% showing a quite fast convergence to the global minimum. The RDO is performed using a number of DPSO function evaluations set to $128 \cdot N_{DV}$, the same as Problem I and II. However, it should be noted that the overall number of function evaluation for the RDO procedure is $128 \cdot N_{DV} \cdot N_{UQ}$, since each evaluation of $\mu$ requires 33 evaluations of the total
resistance.

Figure 5: RDO, UQ and DPSO convergence

Figure 6: Comparison of deterministic optimization versus RDO, optimal design variables and resistance curve

Figure 6a shows the design variables values obtained performing the RDO compared to the deterministic DPSO values of Problem I and II. The RDO optimum falls nearby the optimum of Problem II.

Table 3 summarizes the results of the RDO optimization and, furthermore, shows the values of the expect value of the total resistance evaluated ex-post for the optimal configurations of Problem I and II. Finally, a parametric analysis of the total resistance versus the cruise speed is presented in Fig. 6b, comparing the performances of the optimal RDO configuration and of the optimal configurations of Problem I and II to the original ones. A close agreement is found between Problem II solution and the RDO solution, which however present the best overall improvement in the speed range (Tab. 3). Furthermore, although the RDO solution presents
the overall best performance, Problem I solution is slightly better than the others for v = 10 kn. It should be also noted that the RDO solution outperforms Problem II solution for v = 14 kn, revealing that probably none of the three algorithms had completely explored the search space while solving Problem II.

Table 3: Comparison of deterministic optimization versus RDO, summary of results

<table>
<thead>
<tr>
<th>ID</th>
<th>x₁</th>
<th>x₂</th>
<th>Δµ%</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>0.000</td>
<td>0.000</td>
<td>-</td>
</tr>
<tr>
<td>DPSO Problem I</td>
<td>-1.000</td>
<td>-0.159</td>
<td>11.14</td>
</tr>
<tr>
<td>DPSO Problem II</td>
<td>0.911</td>
<td>-0.503</td>
<td>-2.39</td>
</tr>
<tr>
<td>DPSO RDO</td>
<td>0.554</td>
<td>-0.580</td>
<td>-7.62</td>
</tr>
</tbody>
</table>

5 CONCLUDING REMARKS AND FUTURE WORK

An exploratory comparative study was presented on the performances of three global derivative-free optimization algorithms for simulation-based shape design optimization of hydrodynamic applications. Specifically, the ability of DPSO, DPO and DIRECT was studied for the minimization of the total resistance of a luxury yacht. Two separate deterministic cruise conditions (Problem I and II) were addressed. A stochastic variation of the cruise speed (uniformly distributed within the speed range) was also considered for a RDO problem. The RDO was performed using the best performing algorithm identified from Problem I and II. Two design variables controlled global shape modifications, an in-house potential solver was used to evaluate the hydrodynamic performances of the yacht, and a quasi MC method with a deterministic version of LHS was used to solve the UQ within RDO.

The three algorithms showed overall similar performances, achieving for both deterministic problems comparable reductions of the objective function. For Problem I, DPSO and DPO converged to the minimum faster than DIRECT, whereas for Problem II DPSO showed a faster convergence than the other algorithms. Overall, DPSO was identified as the best performing algorithm and used for the RDO problem.

The deterministic and robust optimization results were compared in terms of total resistance and its expected value. A parametric analysis over the speed range revealed that the RDO solution presents the overall best performance. Moreover, Problem I optimal configuration shows the best performance for its optimization speed (v = 10kn) whereas its performances decrease in the high speed range. Differently, RDO solution is found outperforming Problem II solution at Problem II optimization speed (v = 14 kn), thus revealing that probably during the deterministic optimization the design space was not completely explored. Furthermore, the ex-post evaluation of the expected value of the total resistance of the deterministic solutions showed that Problem I optimal solution has the worst behavior, whereas the deterministic optimal solution of Problem II presents a similar trend to the RDO solution. Future work will extend this study addressing higher-dimensional problems using higher-fidelity hydrodynamic solvers, taking into account diverse probability distributions of cruise speed.
ACKNOWLEDGEMENTS

The research has been partially supported by A. Vallicelli & C. - A.V.C. srl, through contract no. 03CT14, and by the Italian Flagship Project RITMARE, coordinated by the Italian National Research Council and founded by the Italian Ministry of Education.

REFERENCES


MISSION-BASED HULL-FORM AND PROPELLER OPTIMIZATION OF A TRANSOM STERN DESTROYER FOR BEST PERFORMANCE IN THE SEA ENVIRONMENT

G.J. GRIGOROPOULOS*, E.F. CAMPANA†, M. DIEZ†, A. SERANI†, O. GÖREN‡, K. SARIÖZ‡, D.B. DANIŞMAN‡, M. VISONNEAU‡, P. QUEUTEY‡, M. ABDEL-MAKSOUÐ§ AND F. STERN♭

* National Technical University of Athens, School of Naval Engineering, Athens, Greece
e-mail: gregory@central.ntua.gr
†CNR-INSEAN, Natl. Research Council- Marine Technology Research Institute, Rome, Italy
e-mail: {emiliofortunato.campana; matteo.diez}@cnr.it; andrea.serani@insean.cnr.it
‡Istanbul Technical University, Dept. of Naval Architecture and Marine Eng., Istanbul, Turkey
e-mail: {ogoren; sarioz; bulentd}@itu.edu.tr
§Ecole Centrale de Nantes (ECN-CNRS), LHEEA Lab., Nantes, France
e-mail: {michel.visonneau; patrick.queutey}@ec-nantes.fr
♭Hamburg University of Tech., Inst. for Fluid Dynamics and Ship Theory, Hamburg, Germany
e-mail: m.abdel-maksoud@tuhh.de
♭The University of Iowa, IIHR-Hydroscience and Engineering, Iowa City, Iowa, USA
e-mail: frederick-stern@uiowa.edu

Key words: Simulation-based design optimization, Hydrodynamic optimization, Hull-form optimization, Propeller optimization

Abstract. An overview is presented of the activities conducted within the NATO STO Task Group AVT-204 to “Assess the Ability to Optimize Hull Forms of Sea Vehicles for the Best Performance in a Sea Environment.” The objective is the development of a greater understanding of the potential and limitations of the hydrodynamic optimization tools. These include low- and high-fidelity solvers, automatic shape modification methods, and multi-objective optimization algorithms, and are limited here to a deterministic application. The approach includes simulation-based design optimization methods from different research teams. Analysis tools include potential flow and Reynolds-averaged Navier-Stokes equation solvers. Design modification tools include global modification functions, control point based methods, and parametric modelling by hull sections and basic curves. Optimization algorithms include particle swarm optimization, sequential quadratic programming, genetic and evolutionary algorithms. The application is the hull-form and propeller optimization of the DTMB 5415 model for significant conditions, based on actual missions at sea.
1 INTRODUCTION

In order to reduce costs and improve the performance for a variety of missions, navies are demanding new concepts and multi-criteria optimized ships. In order to address this challenge, research teams have developed simulation-based design optimization (SBDO) methods, to generate hull variants and optimize their hydrodynamic performance, combining low- and high-fidelity solvers, design modification tools, and single/multi-objective optimization algorithms. The NATO RTO Task Group AVT-204, formed to “Assess the Ability to Optimize Hull Forms of Sea Vehicles for Best Performance in a Sea Environment,” [1] addressed the integration and assessment of different computational methods and SBDO approaches, bringing together teams from France (ECN-CNRS, Ecole Centrale de Nantes), Germany (TUHH, Hamburg University of Technology), Greece (NTUA, National Technical University of Athens), Italy (CNR-INSEAN, National Research Council-Marine Technology Research Institute), Turkey (ITU, Istanbul Technical University), and United States (UI, University of Iowa).

The objective is the development of a greater understanding of the potential and limitations of the hydrodynamic optimization tools and their integration within SBDO. The former include automatic shape modification tools, low- and high-fidelity solvers, and multi-objective optimization algorithms, and are limited in the present activity to deterministic applications.

The approach encompasses SBDO methods from different research teams, which are assessed and compared. INSEAN and UI undertook a joint effort for a two phase SBDO, using low-fidelity solvers in the first phase, and more accurate and computationally expensive high-fidelity solvers in the second phase. ITU and NTUA performed separate SBDO procedures, based on low-fidelity solvers, whereas ECN-CNRS used a high-fidelity solver to verify low-fidelity optimization outcomes. TUHH addressed the propulsion optimization for the unsteady wake field produced by the optimized hull form. SBDO tools and results are presented in the following, for each research team separately. Analysis tools used in the current study include potential flow (INSEAN/UI, ITU, NTUA, TUHH) and RANS (ECN-CNRS, INSEAN/UI) solvers. Design modification tools include linear expansion of orthogonal basis functions (INSEAN/UI), an approach based on relaxation coefficients at control points with Akima’s surface generation (ITU), and the parametric modelling of the CAESES/FRIENDSHIP-Framework, using a set of basic curves, with associated topological information (NTUA). Multi-objective optimization algorithms include a multi-objective extension of the deterministic particle swarm optimization (INSEAN/UI), a sequential quadratic programming method, which is applied to an artificial neural network model of aggregate objective functions (ITU), and a non-dominated sorting genetic algorithm (NTUA). TUHH modifies the propeller design parameters using a genetic algorithm from Sandia’s Dakota tool kit. The methods and implementations of the optimization procedures were also presented in [1, 2].

The test case for the current study is a deterministic hull-form and propeller optimization of a USS Arleigh Burke-class destroyer, namely the DDG-51 (Fig. 1). The DTMB 5415 model, an open-to-public early concept of the DDG-51, is used for the current research. This has been largely investigated through towing tank experiments [3], and used for earlier SBDO research for conventional/hybrid hulls [4]. Both 5415 bare hull (INSEAN/UI, ECN-CNRS) and the 5415M variant with skeg only (ITU, NTUA) are addressed. The design optimization exercise aims at the reduction of two objective functions, namely (i) the weighted sum of the total resistance in
calm water at 18 and 30 kn (corresponding to Fr=0.25 and 0.41), and (ii) a seakeeping merit factor based on the vertical acceleration at the bridge (in head wave, sea state 5, Fr=0.41) and the roll motion (in stern wave, sea state 5, Fr=0.25). The first speed for resistance optimization (18 kn) is close to the peak of the speed-time profile for transits, from 2013 data [5]. The second speed (30 kn) is the flank speed, used as an objective to minimize the maximum powering requirements. The seakeeping merit factor is based on a first quite extreme condition, and on a second less extreme condition. Sea state 5 is considered, as a commonly encountered open ocean condition for North Atlantic and North Pacific, year round [6]. Although deterministic, the present conditions are a reasonable representation of the operations of a DDG-51.

An early version of this paper was presented in [2], focusing on the low-fidelity hull-form optimization and preliminary validation with RANS. These are extended here, where a complete validation by RANS is presented, along with the propeller optimization and a final RANS-based hull-form optimization.

2 HULL-FORM OPTIMIZATION PROBLEM

The main particulars of the full scale model and test conditions are summarized in Tab. 1. The optimization aims at improving both calm-water and seakeeping performances, and is formulated as

\[
\begin{align*}
& \text{Minimize } \{f_1(\mathbf{x}), f_2(\mathbf{x})\}^T \\
& \text{subject to } g_k(\mathbf{x}) = 0, \quad k = 1, \ldots, K \\
& \text{and to } h_l(\mathbf{x}) \leq 0, \quad l = 1, \ldots, L
\end{align*}
\]

where \(\mathbf{x}\) is the design variable vector, the first objective \(f_1\) is the weighted sum of the normalized total resistance in calm water at 18 kn (Fr=0.25) and 30 kn (Fr=0.41), respectively,

\[
f_1(\mathbf{x}) = 0.85 \frac{R_T}{R_{T_0}}^{18\text{kn}} + 0.15 \frac{R_T}{R_{T_0}}^{30\text{kn}}
\]

with \(R_{T_0}\) the total resistance of the parent hull. This formulation is based on the expertise of the members and some statistical data from US Navy, that destroyers operate most of their time at the fleet speed (close to 18 kn) and only 15% at the maximum speed (around 30 kn). The second objective \(f_2\) is a seakeeping merit factor, defined as

\[
f(\mathbf{x}) = 0.5 \frac{\text{RMS}(a_z)}{\text{RMS}(a_{zo})}^{30\text{kn}} + 0.5 \frac{\text{RMS}(\varphi)}{\text{RMS}(\varphi_0)}^{18\text{kn}}
\]

where RMS represents the root mean square, \(a_z\) is the vertical acceleration at the bridge (located 27 m forward amidships and 24.75 m above keel) at 30 kn in head long-crested waves (180 deg), and \(\varphi\) is the roll angle at 18 kn in stern long-crested waves (30 deg). The wave conditions corresponds to sea state 5, using the Bretschneider spectrum with a significant wave height of 3.25 m and modal period of 9.7 s. Subscript ‘0’ refers to parent-hull values.

The selected dynamic responses are critical at completely different operating conditions (speed, heading, location along the vessel). However, we assume that similar sea conditions prevail in both cases, which form the seakeeping objective. The contribution of each operating condition in the objective is the same (50%).

85
Geometrical equality constraints, \( g_k(x) \), include fixed length between perpendicular and displacement, whereas geometrical inequality constraints, \( h_l(x) \), include limited variation of beam and draught, \( \pm 5\% \), and reserved volume for the sonar in the dome, corresponding to 4.9 m diameter and 1.7 m length (cylinder).

### Table 1: DTMB 5415 model main particulars (full scale)

<table>
<thead>
<tr>
<th>Description</th>
<th>Symbol</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacement</td>
<td>( \nabla )</td>
<td>tonnes</td>
<td>8,636</td>
</tr>
<tr>
<td>Length between perpendiculars</td>
<td>( LBP )</td>
<td>m</td>
<td>142.0</td>
</tr>
<tr>
<td>Beam</td>
<td>( B )</td>
<td>m</td>
<td>18.90</td>
</tr>
<tr>
<td>Draft</td>
<td>( T )</td>
<td>m</td>
<td>6.160</td>
</tr>
<tr>
<td>Longitudinal center of gravity</td>
<td>LCG</td>
<td>m</td>
<td>71.60</td>
</tr>
<tr>
<td>Vertical center of gravity</td>
<td>VCG</td>
<td>m</td>
<td>1.390</td>
</tr>
<tr>
<td>Roll radius of gyration</td>
<td>( K_{xx} )</td>
<td></td>
<td>( 0.40B )</td>
</tr>
<tr>
<td>Pitch radius of gyration</td>
<td>( K_{yy} )</td>
<td></td>
<td>( 0.25LBP )</td>
</tr>
<tr>
<td>Yaw radius of gyration</td>
<td>( K_{zz} )</td>
<td></td>
<td>( 0.25LBP )</td>
</tr>
<tr>
<td>Speed</td>
<td>( U )</td>
<td>kn</td>
<td>{18;30}</td>
</tr>
<tr>
<td>Froude number</td>
<td>( Fr )</td>
<td></td>
<td>{0.25;0.41}</td>
</tr>
<tr>
<td>Water density</td>
<td>( \rho )</td>
<td>kg/m³</td>
<td>998.5</td>
</tr>
<tr>
<td>Kinematic viscosity</td>
<td>( \nu )</td>
<td>m²/s</td>
<td>( 1.09 \cdot 10^{-6} )</td>
</tr>
<tr>
<td>Gravity acceleration</td>
<td>( g )</td>
<td>m/s²</td>
<td>9.803</td>
</tr>
</tbody>
</table>

3 HULL-FORM OPTIMIZATION PROCEDURES AND RESULTS

Three partners, INSEAN/UI, ITU and NTUA undertook independently the optimization task, while ECN-CNRS was responsible for the evaluation of the three derived optimized hull forms and the selection of the most promising one. INSEAN/UI carried out additional CFD calculations, whereas TUHH performed the assessment and optimization of the propeller for the optimal hull form. The following subsection presents briefly the main details of the procedures and the results obtained by INSEAN/UI, ITU, and NTUA, whereas the RANS verification, propeller optimization and additional hull form optimization based on RANS are presented in sections 4, 5, and 6, respectively. Further details may be found in [1, 2].

#### 3.1 INSEAN/UI

The SBDO framework used for the first optimization phase by INSEAN/UI integrates low-fidelity solvers of calm-water resistance and seakeeping prediction, a design modification method based on linear expansion of orthogonal basis function \[7\], and single/multi-objective optimization algorithm based on the particle swarm metaheuristic \[8, 9\]. The WAve Resistance Program (WARP), a linear potential flow code (in-house developed at INSEAN) is used for the calm-water prediction. Details of equations, numerical implementations, and validation of the numerical solver are given in \[10\]. The Standard Ship Motion program (SMP), developed at the David Taylor Naval Ship Research and Development Center, a potential flow solver based on linearized strip theory, is used for the seakeeping prediction \[11\].

The comparison of WARP and SMP with EFD data for the original DTMB 5415 have shown a reasonable agreement. Grid studies have been also performed. Six design spaces are investigated varying the space dimension (with dimensionality ranging from two to six) and the associated design variables bounds. The design space is defined using orthogonal modification functions.

![Figure 1: USS Arleigh Burke Bravo Sea Trials Gulf Off Maine (U.S. Navy photo)](image)
Sensitivity analysis are performed for resistance and motions, showing a significant variability of the performance. The same design spaces are used for single-objective optimization for separate $f_1$ and $f_2$, achieving an improvement by nearly 12% and 13.3% respectively. Multi-objective optimization combining $f_1$ and $f_2$ is finally performed. The most promising design produces an improvement of nearly 7% for both, $f_1$ and $f_2$ and is selected for further investigation by RANS.

### 3.2 ITU

ITU uses a relatively simpler approach in obtaining design modifications (experimental space). The hull form variation is based on a limited number of control points, laying on specific (two in the demo case) waterlines and stations (six in the demo case) along the hull. On these control points relaxation coefficients $1 \pm 0.05$ are applied to deform the hull form and to generate variants. Each variant is then faired using Akima’s method [12]. On the basis of the generated data base of 250 modified hull forms a static Artificial Neural Network (ANN) is trained and the combined (or aggregate) objective function is expressed as $F_{\text{Combined}}^W = w f_1 + (1 - w) f_2$ where $w = \{0, 0.1, \ldots, 0.9, 1.0\}$ is employed as a weighting factor. The selected optimization algorithm is based on sequential quadratic programming (SQP) within Matlab optimization toolbox, suitable for constraint optimization problems whose design variables include upper and lower bounds. The optimal forms for each weighting factor $w$ are investigated by considering $F_{\text{Combined}}^W$ in the ANN training process. The SQP application on the metamodel provided by the ANN gives an optimal point, which is expected to be part of the overall Pareto front. Using the above methodology and numerical analysis by low-fidelity solvers (ITU-Dawson and ITU-SHIPMO for calm-water and seekeeping, respectively) point out 7% and 13.5% improvements in resistance and seakeeping performances, respectively, attained by the selected optimal hull.

### 3.3 NTUA

The optimization is based on the parametric representation of the hull form using CAESES/FRIENDSHIP-Framework. The design is split into a set of surfaces and a total number of ten design variables were selected for hull variation. Five of them refer to the main hull and the rest to the sonar dome. For the hydrodynamic evaluation of the parent and the variant hull forms SWAN2 and SPP-86 potential flow codes are used. The multi-objective optimization with respect to Eq. 1 is carried out by employing the Non-dominating Sorting Genetic Algorithm-II (NSGA-II, [13])]. Parametric modeling using B- and F-Splines, variation and optimization is integrated in CAESES/FFW environment. The hydrodynamic evaluation of the variant hull forms was carried out via the aforementioned codes called within the same environment. The methodology after the generation and the evaluation of 400 faired variants concludes with a Pareto front offering optimized hull forms with varying improvements in resistance and seakeeping. In this case, the improvement in the former results in deterioration of the latter. The selected optimized hull form constitutes a compromise over the two selection criteria, which takes into consideration the magnitude of the improvement in each criterion and its significance on the overall performance of the vessel. The finally proposed optimum hull form offers resistance index reduced by 17% and seakeeping index reduced by 6% over the parent one.
4 VALIDATION OF OPTIMIZED HULL FORMS USING HIGH-FIDELITY SIMULATIONS

The objective of the work by ECN-CNRS is to validate the optimized geometries (full scale) with their in-house ISIS-CFD code [14, 15, 16]. The flow solver, available as a part of the FINE™/Marine computing suite, is an incompressible unsteady Reynolds-averaged Navier-Stokes (URANS) method mainly devoted to marine hydrodynamics (a typical domain is shown in Fig. 2).

ECN-CNRS evaluated two designs proposed by INSEAN, the NK-WC (obtained by Neumann-Kelvin linearization with the transverse Wave Cut method [17]) and DM-PI (obtained by Double-Model [18] Pressure Integral method), and the designs proposed by ITU and NTUA (Fig. 3). The latest was found significantly better than the other three with respect to the present rating criteria. Furthermore, the calm water resistance reduction of the optimized over the parent hull form as predicted by SWAN2 2002 software, a potential flow method incorporated in NTUAs methodology, is quite similar to the one derived using the URANS method. The other three potential flow methods overestimate the performance improvement of the optimized hull. The NTUA geometry offers a 6.1% reduction for $f_1$, and specifically a 8.8% reduction of total resistance at Fr=0.25, with a 3% increment at Fr=0.41.

Table 2: Optimization results summary: RANS validation of PF-based optimal solutions

<table>
<thead>
<tr>
<th>Geometry</th>
<th>$C_T \times 10^{-3}$ [Fr = 0.25]</th>
<th>$C_T \times 10^{-3}$ [Fr = 0.41]</th>
<th>RMS($\alpha_z$) [m/s²]</th>
<th>RMS($\phi$) [rad]</th>
<th>$\Delta f_1$%</th>
<th>$\Delta f_2$%</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>2.702</td>
<td>4.960</td>
<td>-</td>
<td>1.296</td>
<td>0.018</td>
<td>-</td>
</tr>
<tr>
<td>INSEAN/UI</td>
<td>3.018</td>
<td>5.314</td>
<td>9.3</td>
<td>1.254</td>
<td>0.020</td>
<td>3.1</td>
</tr>
<tr>
<td>NTUA</td>
<td>2.435</td>
<td>5.112</td>
<td>-6.1</td>
<td>1.314</td>
<td>0.019</td>
<td>0.7</td>
</tr>
<tr>
<td>ITU</td>
<td>2.801</td>
<td>5.043</td>
<td>6.2</td>
<td>1.275</td>
<td>0.019</td>
<td>1.2</td>
</tr>
</tbody>
</table>

(a) INSEAN/UI (NK-WC)  (b) ITU  (c) NTUA

Figure 3: Optimized and original hull stations
The seakeeping calculations using URANS code were limited and the achieved changes on the vertical dynamic responses were quite limited.

Figures 4 and 5 show the wave elevation patterns evaluated by ISIS-CFD at Fr=0.28 (used for comparison with experimental data available for the original hull) and Fr=0.41, respectively. A summary of the results is presented in Tab. 2.

(a) INSEAN/UI (NK-WC)
(b) ITU
(c) NTUA

Figure 4: Optimized and original wave elevations distribution for calm water at Fr=0.28

(a) INSEAN/UI (NK-WC)
(b) ITU
(c) NTUA

Figure 5: Optimized and original wave elevations distribution for calm water at Fr=0.41

5 PROPELLER OPTIMIZATION FOR OPTIMIZED HULL FORM

TUHH team used the in-house boundary element solver panMARE (a typical grid is shown in Fig. 6) and the wake field provided by ECN-CNRS CFD-code ISIS to design the propeller. The latter always works in an inhomogeneous wake field due to the presence of the ship’s hull in front of the propeller. This wake field is a major design factor for the propeller. Although the propeller is usually designed on the basis of the calm water condition, it has to cope with the real operation conditions where the wake field is unsteady due to the incoming waves and the ship motions. The unsteady wake field can be a reason for increasing the power demand and the pressure fluctuation amplitudes. Within this study advanced optimization algorithms are used to design propellers with improved characteristics in seaways. The varying wake field is considered at four cases within a cycle of a regular wave, propeller on the crest or the trough and at a wave node moving up- or downwards.

The variant propeller designs were evaluated in two stages. The aim of the first stage is to develop geometries, which satisfy the demand for averaged delivered thrust with a minimum
value of required torque. In a second stage, a number of most successful designs are investigated regarding their cavitation behavior.

The goal of this study is to design an optimal propeller for an unsteady operating conditions. After a number of simplifications, a numerical setup is achieved that allows for considering the most important physical aspects of the problem. The setup is used to evaluate the individuals generated within a defined search space. The number of individuals which satisfy the imposed constraint is gradually increased with the evolutionary progress of the optimization. In addition, the propeller efficiency also shows a considerable improvement. The shape of the optimized propeller and the pressure distribution are shown in Fig. 7. The analysis of the optimized geometries shows that the pitch is reduced at the tip to limit the tip vortex circulation. Accordingly, a slight increase in camber is present. For mid-section parameters, a slow convergence behavior has been observed. The influence of the evolutionary algorithm settings on the results should therefore be studied with regard to accelerating the convergence while retaining the converging character. In a second simulation stage, the best individuals from the evolutionary run are evaluated concerning cavitation probability. Many geometries show little thin cavitating line near the leading edge in the investigated operating conditions. Further numerical studies based on RANS-simulations are needed to check whether phenomena like leading edge vortex may be the reason for local separation in this area. Details may be found in [1].

6 EFFECTS OF LOW- AND HIGH-FIDELITY SOLVERS ON HULL-FORM OPTIMIZATION

High-fidelity solvers (such as RANS) have shown their capability to provide accurate solutions to the design problem [19]. Their computational cost is still a critical issue in SBDO. For this reason, metamodels and variable-fidelity approaches, based on low- and high-fidelity solvers, have been developed and applied to reduce the computational time and cost of the SBDO. Low-fidelity solvers (such as potential flow, PF) have been applied to identify suitable design spaces for RANS-based optimization. Identifying the proper trend of the design objective versus the design variables often represents a critical issue for a low-fidelity solver, especially when large design modifications are involved. The choice of a low-fidelity solver within SBDO represents a critical issue and should be carefully justified, considering the trade-off between computational efficiency and solution accuracy.

INSEAN/UI compared the effects of four PF formulations and implementations on the results of the multi-objective SBDO problem in Eq. 1. Kelvin and Dawson linearization (referred to as Neumann-Kelvin, NK, and double model, DM, respectively) are used with a standard pressure integral over the body surface (referred to as PI) and the transverse wave cut method (referred to as WC), for the wave resistance calculation. A sensitivity analysis at Fr=0.25 using RANS
is shown in [1], for comparison and correlation with the potential flow solutions. The code WARP was used as PF solver, whereas the RANS computations were performed with the code CFDShip-Iowa [20, 21]. It was found that the PF formulation significantly affects the SBDO outcomes. Specifically, the Pareto fronts look quite different and the selected optimal designs fall in different region of the design space, depending on the PF formulation used. The following considerations can be made [1]: the validation for the original hull shows reasonable trends, but NK-PI for low Fr; DM shows better validation especially for sinkage, compared to NK; NK-PI provides significant resistance reductions at low Fr (likely due to an overestimate of the resistance for the original hull) and more limited improvements at high Fr; NK-WC shows a quite opposite trend; DM-PI indicates more limited (and realistic) improvements, for both low and high Fr; it also shows a limited possibility of improving both objectives at the same time; DM-WC provides more significant resistance reduction at high Fr; overall, the WC method always indicates greater improvements at high Fr than PI, likely due to an overestimate for the resistance of the original hull; NK results seem more affected by the wave resistance estimation method than DM.

![Hull stations and wave elevation pattern](Figure 8: Comparison of optimized and original hull stations and wave elevation distribution at Fr=0.25)

These outcomes motivated further investigations by RANS. Specifically, a sensitivity analysis at Fr=0.25 was conducted and compared with the PF results, showing several differences between PF and RANS solutions. Specifically, none of the PF formulations showed a reasonable trend for all the design variables, compared to RANS. The analysis of the Pearsons correlation coefficient between PF and RANS results showed a good correlation between NK-PI and RANS for four out of six variables. For the current application, NK-PI is the more effective PF formulation.

Table 3: Comparison between original and IN-SEAN/UI RANS-optimized DTMB 5415 hydrodynamic coefficients

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Unit</th>
<th>Original</th>
<th>Optimized</th>
<th>∆%</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_{pp}$</td>
<td></td>
<td>1.38E-03</td>
<td>9.08E-04</td>
<td>-34.0</td>
</tr>
<tr>
<td>$C_h$</td>
<td></td>
<td>0.86E-03</td>
<td>1.24E-03</td>
<td>42.0</td>
</tr>
<tr>
<td>$C_f$</td>
<td></td>
<td>3.16E-03</td>
<td>3.18E-03</td>
<td>0.65</td>
</tr>
<tr>
<td>$C_{mg,x}$</td>
<td></td>
<td>-1.19E-03</td>
<td>1.35E-03</td>
<td>-13.4</td>
</tr>
<tr>
<td>$C_T$</td>
<td></td>
<td>4.21E-03</td>
<td>3.97E-03</td>
<td>-6.00</td>
</tr>
<tr>
<td>$\sigma/LBP$</td>
<td>deg</td>
<td>-1.31E-03</td>
<td>-1.35E-03</td>
<td>-3.29</td>
</tr>
<tr>
<td>$\tau$</td>
<td>deg</td>
<td>-0.11</td>
<td>-0.12</td>
<td>-15.3</td>
</tr>
<tr>
<td>$S_{w,stat}/LBP^2$</td>
<td></td>
<td>1.48E-02</td>
<td>1.50E-02</td>
<td>0.96</td>
</tr>
</tbody>
</table>

Since low-fidelity solvers can lead to inaccurate design solutions (especially for large design modifications and possible flow separation), a further RANS-based optimization of the DTMB 5415 bare hull was proposed. Specifically, a deterministic derivative-free single-objective optimization was performed, using global/local hybridization by derivative-free line search methods of two well-known global algorithms, DIRECT and DPSO, respectively [1, 7]. The optimization
aimed at the reduction of the (model scale) total resistance coefficient in calm water at Fr=0.25. The design space was generated by a linear expansion of orthogonal basis functions for the modification of the hull form. The problem was solved with a number of design variables equal to eleven. A resistance reduction of 6% was achieved by the optimized design. The final shape obtained with RANS induces a high pressure region in correspondence of the first trough of the diverging bow wave of the original hull. This causes a phase shift with a significant reduction of the bow wave and the cancellation of the shoulder wave. As a result, the pressure distribution appears more uniformly distributed along the hull and most of the resistance reduction stems from the piezometric pressure coefficient. The final hydrodynamic assessment of the RANS-based optimized shape has confirmed the effectiveness of the SBDO procedure, driven by hybrid global/local methods. Results are shown in Tab. 3 and Fig. 8.

7 CONCLUSIONS

A multi-objective hull form and propeller optimization of the DTMB 5415 (specifically the MARIN variant 5415M, with skeg only) was investigated using low- and high-fidelity solvers, performed by different research team (INSEAN/UI, ITU, NTUA, ECN-CNRS, and TUHH). Overall, optimization achievements by low-fidelity solvers were found significant, with an average improvement for calm-water resistance and seakeeping performances of 10 and 9% respectively. The most promising designs show up to 16% improvement for the calm-water resistance and 14% for the seakeeping merit factor. The design-space size ranged from two to twelve and the optimized designs show a quite large variability and different characteristic.

INSEAN/UI defined six design spaces with dimensionality ranging from two to six, using a linear expansion of orthogonal basis functions for the modification of the DTMB 5415 bare hull. The optimization was performed by a multi-objective extension of the deterministic particle swarm optimization algorithm. ITU produced 250 hull form variants of the 5415M using Akima’s surface generation, with randomly distributed relaxation coefficients at control points over the body surface. The optimization procedure combined an artificial neural network with a sequential quadratic programming algorithm, which is fed with aggregate objective functions. NTUA used the parametric modelling of the CAESES/FRIENDSHIP-Framework for the design modification of the 5415M, representing the hull form by a set of basic curves, providing topological information, and defining a set of 19 sections. The hull surface was parametrized by ten design variables. The NSGA II code was used for the optimization procedure. ECN-CNRS verified parent and optimal hulls, using an in-house high-fidelity solver (ISIS-CFD). The geometry provided by NTUA was selected as the best candidate, providing a 6.1% reduction of the calm water resistance (weighted average at Fr=0.25 and 0.41). TUHH performed the propeller optimization considering the unsteady wake field in waves. Finally, further investigations on the effects of potential flow formulation/linearization on the multi-objective optimization were proposed by INSEAN/UI, along with a RANS-based optimization.

The methodologies proposed have been found a viable option for SBDO. Low-fidelity solvers have shown some limitations in the prediction of the objective trends (especially for the resistance). High-fidelity solvers should be used, whenever possible. SBDO techniques are mature for extension to more complex aspects of the hydrodynamics of naval combatants (maneuvering, intact and dynamic stability, etc.) as well as other items/disciplines (structures, operations,
economic management, weight, etc.). Moving to more complex, real-world, multi-disciplinary problems [22], particular attention should be paid to the trade-off between computational accuracy and cost, and the interplay among the different elements and disciplines involved. Finally, SBDO research would benefit from experimental fluid dynamics (EFD) of original and optimized designs, whenever possible.

ACKNOWLEDGEMENTS

The work has been performed in collaboration with NATO RTO Task Group AVT-204 “Assess the Ability to Optimize Hull Forms of Sea Vehicles for Best Performance in a Sea Environment.” CNR-INSEAN and University of Iowa teams are grateful to Dr. Thomas Fu, Dr. Woei-Min Lin, and Dr. Ki-Han Kim of the US Navy Office of Naval Research, for their support through NICOP grant N62909-15-1-2016 and grant N00014-14-1-0195. INSEAN team is also grateful to the Italian Flagship Project RITMARE, coordinated by the Italian National Research Council and funded by the Italian Ministry of Education. ECN/CNRS team gratefully acknowledge GENCI (Grand Equipement National de Calcul Intensif) for the HPC resources of IDRIS and CINES under Grant 21308.

REFERENCES


93


MULTI-OBJECTIVE HULL-FORM OPTIMIZATION OF A SWATH CONFIGURATION VIA DESIGN-SPACE DIMENSIONALITY REDUCTION, MULTI-FIDELITY METAMODELS, AND SWARM INTELLIGENCE

RICCARDO PELLEGRINI*, ANDREA SERANI*, STEFAN HARRIES† AND MATTEO DIEZ*

*CNR-INSEAN, National Research Council-Marine Technology Research Institute, Rome, Italy
e-mail: matteo.diez@cnr.it
†Friendship Systems AG, Potsdam, Germany
e-mail: harries@friendship-systems.com

Key words: Simulation-based design, Hull-form optimization, Design-space dimensionality reduction, Karhunen-Loève expansion, Multi-fidelity metamodels, Multi-objective particle swarm optimization

Abstract. A multi-objective simulation-based design optimization (SBDO) is presented for the resistance reduction and displacement increase of a small water-plane area twin hull (SWATH). The geometry is realized as a parametric model with the CAESES® software, using 27 design parameters. Sobol sampling is used to realize design variations of the original geometry and provide data to the design-space dimensionality reduction method by Karhunen-Loève expansion. The hydrodynamic performance is evaluated with the potential flow code WARP, which is used to train a multi-fidelity metamodel through an adaptive sampling procedure based on prediction uncertainty. Two fidelity levels are used varying the computational grid. Finally, the SWATH is optimized by a multi-objective deterministic version of the particle swarm optimization algorithm. The current SBDO procedure allows for the reduction of the design parameters from 27 to 4, resolving more than the 95% of the original geometric variability. The metamodel is trained by 117 coarse-grid and 27 fine-grid simulations. Finally, significant improvements are identified by the multi-objective algorithm, for both the total resistance and the displacement.

1 INTRODUCTION

Simulation-based design optimization (SBDO) assists the designer in the design process of complex engineering systems (such as aerial, ground, or maritime vehicles). In shape optimization, SBDO combines shape modification methods, the assessment of the design performances, and single or multi-objective optimization algorithms.

The shape modification can be performed by applying modification operators to the original design or defining the geometry through a parametric model [1]. Each parameter may be consid-
ered as a design variable for the optimization process, therefore the number of design variables may easily be significantly large for complex models and geometries. Recent research has focused on research space variability and dimensionality reduction as an essential part of SBDO. A quantitative approach based on the Karhunen-Loève expansion (KLE) has been formulated for a pre-optimization assessment of the shape modification variability and the definition of a reduced-dimensionality global model of the shape design [2].

The SBDO process often requires computationally expensive physics-based solvers to achieve accurate solutions. In order to reduce the computational cost of the SBDO process, metamodels are often used. These have been successfully developed and applied in many engineering fields [3]. Volpi et al. [4] presented a dynamic radial basis functions (RBF) metamodel for ship hydrodynamics problems. Its extension to design optimization has been presented in [5]. Lately, Giselle et al. [6] presented a survey on parallel surrogate-assisted global optimization for expensive functions.

Combining metamodelling methods with multi-fidelity approximations potentially leads to a further reduction of the computational cost. Multi-fidelity approximation methods have been developed with the aim of combining to some extent the accuracy of high-fidelity solvers with the computational cost of low-fidelity solvers [7]. Correction methods, such as additive and/or multiplicative approaches, are used to build multi-fidelity metamodels. High- and low-fidelity models may be determined by the physical model [7], and/or the size of the computational grid [8]. Multi-fidelity metamodels have been used for engineering applications including ships [9, 10].

For engineering applications, global derivative-free optimization algorithms represent an advantageous option for their (often common) ease of implementation and capability of providing adequate solutions to the optimization problem. Among this kind of algorithms, particle swarm optimization (PSO) was originally introduced [11] as a global derivative-free metaheuristics for single-objective optimization. The algorithm makes use of cognitive and social attractors based on individual and population optima, in order to steer the swarm dynamics. PSO has been extended to multi-objective optimization (MOPSO) in [12]. Generally, MOPSO extends the concept of cognitive and social attractors to the multi-objective context, using individual and population Pareto fronts, sub-swarms, or aggregate objective functions. A comprehensive survey on MOPSO variants has been provided in [13]. Most PSO (both single- and multi-objective) formulations include stochastic methods and/or random coefficients. This implies that in order to assess the algorithm performance, statistically significant results need to be produced, through extensive numerical campaigns. Such an approach is often too expensive (from the computational viewpoint) and therefore not practicable in SBDO (especially when computationally expensive solvers are used directly). For this reason, efficient deterministic approaches, namely deterministic PSO (DPSO) [14] and multi-objective deterministic PSO (MODPSO) [15] have been developed and successfully applied in SBDO.

The objective of the present work is the application and preliminary assessment of a SBDO methodology, based on design space dimensionality reduction, adaptive multi-fidelity metamodel (AMFM), and multi-objective deterministic particle swarm optimization, to a 36.5 m small water-plane area twin hull (SWATH) configuration.

The parametric geometry of the SWATH is produced with the computer-aided design (CAD) environment integrated in CAESER®, developed by FRIENDSHIP SYSTEMS. Subsequently, the design-space dimensionality reduction of the parametric model is performed by KLE [2].
The hydrodynamic performance is assessed by the potential flow solver WARP, developed at CNR-INSEAN [16]. The evaluations provided by WARP are used to build the AMFM of the total resistance and the displacement with a coarse and a fine panel grid. Finally, the MODPSO formulation presented in [17] is used for the multi-objective optimization with the aim of reducing the resistance in calm water at 18 kn and increasing the displacement.

2 PROBLEM STATEMENT AND SBDO METHODS

2.1 Model geometry and optimization problem

The SWATH is designed as two torpedoes connected to the upper platform by a couple of twin symmetric narrow struts for each hull (for a total of four struts). The main geometric particulars are: overall length $L_{OA} = 36.50$ m (without propeller/rudder); torpedo maximum diameter $D = 4.50$ m; inter-axis distance $D_{H} = 20.00$ m; first strut leading edge position $L_{1} = 6.00$ m, struts length $S_{L} = 12.00$ m, and struts clearance $S_{C} = 5.15$ m; drought $T = 6.31$ m; wet surface $S_{W} = 1064$ m$^2$ and water-plane area $A_{WP} = 38.88$ m$^2$; displacement $\nabla = 982.23$ m$^3$.

A multi-objective optimization for calm water performances at 18 kn is sought after, as a significant test case for the SBDO methodology. The optimization problem reads

$$\text{minimize } f(x) = \{R_T(x), -\nabla(x)\}^T, \text{ with } x \in D \subset \mathbb{R}^{N_{dv}}$$

subject to $l \leq x \leq u$ (1)

where $x$ is the design variable vector, $R_T$ is the total resistance, $\nabla$ is the ship displacement, $N_{dv}$ is the number of design variables, and finally $l$ and $u$ are the lower and upper bounds for $x$.

The geometry is realized as a parametric model in CAESES®, using a set $x$ of 27 parameters. These parameters define, among other design features, the overall length, the struts clearance, the curvature of the torpedo nose, and the torpedo diameter. The inter-axis distance is held constant. Figure 1 shows the complete SWATH model as produced by CAESES® (a) and a simplified version used for the numerical simulations (b).

Critical design requirements and constraints associated to the modification of the displacement, water-plane area, and geometry of the struts (such as intact pitch and roll stability, seakeeping, and structural analysis) are beyond the scope of the present demonstration and will be addressed in future research.

2.2 Production of design variants through the CAESES® system

CAESES® (CAE System Empowering Simulation) by FRIENDSHIP SYSTEMS AG provides simulation-ready parametric CAD for complex shapes. The embedded CAD environment provides, beside all the most used features for modelling, specific tools for the ship hull and blade (for both propellers and turbo-machinery) design. Other features are provided for helping the designer to assess the quality of the parametric model (such as parameter sensitivity analysis). Furthermore, design modifications are allowed both with pseudo-random variation of design parameters or the free-form deformation techniques.

In the present work, the Sobol engine available in CAESES® is used for producing pseudo-random variations of the 27 geometric parameters. The Sobol method [18] provides a uniform
distribution of the parameters of the design space under investigation, which is used as the input of the design-space dimensionality reduction procedure.

### 2.3 Design-space dimensionality reduction by KLE

The Karhunen-Loève expansion of the shape modification vector produced by the Sobol method provides a pre-optimization (offline) assessment of the shape modification variability and the definition of a reduced-dimensionality global model of the shape modification vector [2]. Similar formulations have been developed and applied to elastic deformation of structures, including modal identification studies [19, 20].

A brief description of the method is recalled in the following. This can be applied with any shape modification method. Consider a geometric domain of interest $G$, which identifies the initial shape, and a set of coordinates $\xi \in G$. Assume that the design variables $x$ defines a shape modification vector $\delta(\xi, x)$. Consider the design variables $x$ as belonging to a stochastic domain $D$ with associated probability density function $p(x)$. The associated mean shape modification is evaluated as

$$\langle \delta \rangle = \int_D \delta(\xi, x)p(x)dx$$

(2)

If one defines the internal product in $G$ as

$$(f, g) = \int_G f(\xi) \cdot g(\xi) \, d\xi$$

(3)

with associated norm $\|f\| = (f, f)^{1/2}$, the variance associated to the shape modification vector (geometric variance) may be defined as

$$\sigma^2 = \left\langle \|\delta\|^2 \right\rangle = \int_D \int_G \delta(\xi, x) \cdot \delta(\xi, x)p(x)d\xi dx$$

(4)

where $\hat{\delta} = \delta - \langle \delta \rangle$ and $\langle \cdot \rangle$ denotes the ensemble average over $x \in D$.

In this context, the aim of the KLE is to find an optimal basis of orthonormal functions, for the linear representation of $\delta$, expressed by

$$\hat{\delta}(\xi) \approx \sum_{k=1}^{N} \alpha_k \phi_k(\xi)$$

(5)
where $\alpha_k$ are the basis-function components, used hereafter as new design variables, and $\{\phi_k\}_{k=1}^{\infty}$ (called KL modes) are the solutions of the eigenproblem

$$L \phi(\xi) = \int G \left( \delta(\xi, x) \otimes \delta(\xi', x) \right) \phi(\xi')d\xi' = \lambda \phi(\xi)$$  \hspace{2cm} (6)

The reduced dimension $N$ is selected in order to retain a prescribed level $l$ (with $0 \leq l \leq 1$) of the original geometric variance. Using the property of the KLE eigenvalues (also called KL values), $N$ in Eq. 5 is selected such as

$$\sum_{k=1}^{N} \lambda_k \geq l \sum_{k=1}^{\infty} \lambda_k = l \sigma^2$$  \hspace{2cm} (7)

with $\lambda_k \geq \lambda_{k+1}$. Details of equations and numerical implementations are given in [2].

### 2.4 Adaptive multi-fidelity metamodel

If one considers $M$ functions of interest (relevant outputs), the multi-fidelity metamodel (MFM) is defined as

$$\hat{f}_i(x) = \tilde{f}_{i,L}(x) + \tilde{\varepsilon}_i(x), \quad \text{with} \quad i = 1, \ldots, M$$

$$\varepsilon_i(x) = f_{i,H}(x) - f_{i,L}(x), \quad \text{with} \quad i = 1, \ldots, M$$  \hspace{2cm} (8)

where superscript “$\sim$” denotes the metamodel prediction, and $\varepsilon_i$ is the difference (error) between high- and low-fidelity simulations ($f_{i,H}$ and $f_{i,L}$, respectively).

The uncertainty associated with the prediction provided by the MFM of the $i$-th function is defined as $U_{\hat{f}_i}(x) = \sqrt{U^2_{\tilde{f}_{i,L}}(x) + U^2_{\tilde{\varepsilon}_i}(x)}$, where $U_{\tilde{f}_{i,L}}$ and $U_{\tilde{\varepsilon}_i}$ are the uncertainties associated to the prediction of the $i$-th function, provided by the low-fidelity and error metamodels ($\tilde{f}_{i,L}$ and $\tilde{\varepsilon}_i$), respectively [10].

![Figure 2: Multi-fidelity metamodel, adaptive sampling procedure](image)

The MFM is trained using the adaptive procedure shown in Fig. 2, resulting in an adaptive MFM (AMFM). After initialization, a new sample is added to the training set at each iteration, solving $(x^*, i^*) = \arg\max_{x,i}[U_{\hat{f}_i}(x)]$. Once $x^*$ and $i^*$ are evaluated, the training sets $H$ and/or $L$ (high- and low-fidelity, respectively) are updated as

$$\begin{cases} 
\text{If } U^2_{\tilde{f}_{i^*,L}}(x^*) \geq \beta U^2_{\tilde{\varepsilon}_{i^*}}(x^*), \text{ then add } x^* \text{ to } L \\
\text{If } U^2_{\tilde{f}_{i^*,L}}(x^*) < \beta U^2_{\tilde{\varepsilon}_{i^*}}(x^*), \text{ then add } x^* \text{ to } H \text{ and } L 
\end{cases}$$  \hspace{2cm} (9)
where \( \beta \in [0, 1] \) is an arbitrary tuning parameter, related to the ratio of the computational cost of low- and high-fidelity simulations.

In the present work, the prediction \( \tilde{f} \) is evaluated as the expected value (EV) of a set of stochastic RBF predictions \([4]\), which depend on the stochastic parameter \( \tau \sim \text{unif}[1, 3] \):

\[
\tilde{f}(x) = EV[g(x, \tau)] \quad \text{with} \quad g(x, \tau) = \sum_{j=1}^{J} w_j \| (x - x_j) \|^\tau
\]  

where \( J \) is the size of the training set, \( x_j \) are the training points, and \( \| \cdot \| \) is the Euclidean norm. The coefficients \( w_j \) are obtained by the linear system \( Aw = y \) with \( w = \{ w_j \} \). The elements of the matrix \( A \) are \( a_{jk} = \| x_j - x_k \|^\tau \) and the vector \( y = \{ y_j \} \) collects the function evaluations at the training points, \( y_j = f(x_j) \).

The uncertainty \( U(x) \) associated to the metamodel prediction is quantified at each \( x \) as the 95%-confidence interval of \( g(x, \tau) \). This is evaluated using a Monte Carlo sampling over \( \tau \), as shown in \([4]\).

### 2.5 Potential flow solver WARP

The WAve Resistance Program (WARP) code, developed by the CNR-INSEAN \([16]\), is used for the numerical solution of the potential flow equations. For the current application, wave resistance computations are based on the linear potential flow theory, with Dawson (or double-model) linearization. The wave resistance is evaluated using a pressure integral over the body surface, whereas the frictional resistance is estimated using a flat-plate approximation based on the local Reynolds number. Sinkage and trim are fixed (even keel).

The computational grids are defined using a refinement ratio of \( \sqrt{2} \). The high- and low-fidelity grids (G1 and G2, respectively) have 5.2k and 2.6k panels for the body and 6k and 3k panels for the free-surface, respectively. Half domain is modelled, using problem symmetry. The computational domain dimensions are \( 1.5L_{OA} \) upstream, \( 3.5L_{OA} \) downstream, and \( 1.5L_{OA} \) sideways. Figure 3 shows the computational grids for the free surface and hull for both G1 (Fig. 3a,b) and G2 (Fig. 3c,d). For the current problem, each high-fidelity simulation requires an average wall-clock time of 4 minutes on an Intel Xeon E5-1620 v2 @3.70GHz, whereas each low-fidelity simulation requires 1 minute. The resulting computational-time ratio is \( \beta = 0.25 \).

![High- and low-fidelity computational grids](image)

**Figure 3:** High- and low-fidelity computational grids

### 2.6 Multi-objective deterministic PSO

PSO algorithm \([11]\) is based on the social-behaviour metaphor of a flock of birds or a swarm of bees searching for food and belongs to the class of metaheuristic algorithms for single-objective
derivative-free global optimization. Pinto et al. [15] proposed a multi-objective deterministic version of PSO (MODPSO) as

\[
\begin{align*}
\mathbf{v}_{i}^{k+1} &= \chi \left[ \mathbf{v}_{i}^{k} + c_1 \left( \mathbf{p}_i - \mathbf{x}_i^{k} \right) + c_2 \left( \mathbf{g}_i - \mathbf{x}_i^{k} \right) \right] \\
\mathbf{x}_i^{k+1} &= \mathbf{x}_i^{k} + \mathbf{v}_i^{k+1}
\end{align*}
\]

(11)

where \( \mathbf{v}_i^{k} \) and \( \mathbf{x}_i^{k} \) are the velocity and the position of the \( i \)-th particle at the \( k \)-th iteration, \( \chi \) is a constriction factor, \( c_1 \) and \( c_2 \) are the cognitive and social learning rate, and \( \mathbf{p}_i \) and \( \mathbf{g}_i \) are the cognitive and social attractor.

The algorithm formulation and setup is defined as suggested in [17]: the cognitive attractor \( \mathbf{p}_i \) is the personal minimizer of the aggregate function \( F(\mathbf{x}_i) = \sum_{m=1}^{M} w_m f_m(\mathbf{x}_i) \), where \( w_m = 1/M \) (\( \forall m \)) is the weight associated to the \( m \)-th objective function with \( M \) the number of objective functions; the social attractor \( \mathbf{g}_i \) is the closest point to the \( i \)-th particle of the Pareto front; the number of particles is set equal to 64 (8MNdv), initialized over domain and boundary with a Hammersley distribution and non-null velocity [21]; the coefficients correspond to \( \chi = 0.721, c_1 = c_2 = 1.655 \) [22]; a semi-elastic wall-type approach [14] is used to keep the particles inside the feasible domain. The budget of problem evaluations is set equal to 16,000 (2000MNdv).

3 NUMERICAL RESULTS

A preliminary assessment of the computational grids (G1 and G2) is performed. Figure 4 shows the total resistance evaluated using G1 and G2, along with the error \( \varepsilon = R_T|G1 - R_T|G2 \) versus the advancing speed.

A number of \( S = 11,500 \) random designs are produced assuming a uniform distribution \( p(\mathbf{x}) \). Figure 5 shows the KLE results in terms of design variability associated to a reduced-dimensionality space of dimension \( N \) for \( S = 3,000, 6,000, \) and 11,500 samples. The results are found convergent versus \( S \). The number of design variables is reduced to \( N = 4 \), retaining the 95% of the original variability. The corresponding KL modes are shown in Fig. 6. For the reduced-dimensionality representation of Eq. 5, modes are normalized such as \( \left( \phi_k, \phi_k \right) = 3\lambda_k \), \( \forall k \), assuming a uniform distribution for \( \alpha_k \). Accordingly, the new design variable range is set to \(-1 < \alpha_k < 1, \forall k \). A preliminary sensitivity analysis is performed along the KL modes for \( R_T \) (Fig. 7a) and \( \nabla \) (Fig. 7b), respectively. It is worth noting that both \( R_T \) and \( \nabla \) are mainly influenced by the first two KL modes.

Sensitivity analysis values are used as initial training set for the AMFM, resulting in 17 high- and low-fidelity analysis. A convergence value for the maximum prediction uncertainty (\( U_{\text{Max}} \)) of the AMFM is set equal to 5% of the initial \( R_T \) and \( \nabla \) values. A maximum budget of 100 iterations is used for the adaptive sampling procedure. Unfeasible configurations are penalized.
Figure 6: KL modes \((k = 1, \ldots, 4)\), represented on the original (unmodified) grid

Figure 7: Sensitivity analysis along KL modes
Figure 8 shows the convergence of $U_{\text{Max}}$ for the prediction of $R_T$ and $\nabla$ during the adaptive training process. Square marks indicate where high-fidelity evaluations are performed, whereas arrows indicate where the penalization is used. The displacement maximum uncertainty decreases more rapidly and with less oscillations than that for the total resistance. The whole budget (100 iterations) is used for training the AMFM, achieving a $U_{\text{Max}}$ is equal to 9.1% and 5.6% for $R_T$ and $\nabla$, respectively. A total of 117 low-fidelity and 27 high-fidelity evaluations are performed, including the initial training set, resulting in a total wall-clock time equal to 225 minutes on an Intel Xeon E5-1620 v2 @3.70GHz.

Figure 9a shows the Pareto front provided by the AMFM based MODPSO and three selected optimal designs: (A) maximum total resistance reduction, (B) maximum displacement increase, and (C) minimum aggregate objective function (with equal weights). The corresponding optimal design variables values are shown in Fig. 9b. Figure 9c shows the total resistance versus the speed for the original and the three optimal designs. All the selected designs perform better than the original SWATH at 18 kn, whereas the original design shows better performances at lower and higher speeds. Table 1 summarizes the main geometric changes ($L_{\text{OA}}$, $S_W$, and $A_{\text{WP}}$) associated to the optimized configurations along with AMFM predictions and actual fine-grid evaluations by WARP.

Finally, Fig. 10a compares the wave elevation of optimal (C) and original designs. The optimized configuration produces a diverging Kelvin wave that is significantly reduced. Figure 10b shows the pressure distribution on the optimal (C) and original hulls. The optimized hull shows lower pressure gradients and a better pressure recovery towards the stern. Figure 10c
shows comparison between the optimized and original hulls. The optimized SWATH is more than 20% longer \((L_{\text{OA}} = 44.4 \, \text{m})\) than the original, whereas the strut length is about 8.4% smaller \((S_L = 11.0 \, \text{m})\).

4 CONCLUSIONS

A multi-objective SBDO of a 36.5 m SWATH has been assessed for the reduction of the total resistance at 18 kn and the increase of the displacement.

The geometry has been realized as a parametric model within CAESES\textsuperscript{®} using 27 design parameters. The Sobol engine available in CAESES\textsuperscript{®} has been used to modify the original geometry and provide data to the design-space dimensionality reduction method by KLE. The original design space has been reduced in dimensionality to 4 variables, while resolving more than the 95% of the original geometric variability associated to the shape modification. The KL modes have been used as a basis to build the new reduced-dimensionality design space.

An adaptive multi-fidelity metamodel has been trained by the potential flow solver WARP using two grids, namely defining the high- and low-fidelity evaluations. The adaptive sampling procedure has required the whole budget of 100 iterations, achieving a prediction uncertainty equal to 9.1% and 5.6% for total resistance and displacement, respectively. Specifically, 27 high- and 117 low-fidelity evaluations have been used. A comparison of metamodel predictions and high-fidelity evaluations has been carried out for selected designs, showing a close agreement.

The optimization has been carried out on the metamodel using a MODPSO algorithm, obtaining a discontinuous and convex/concave Pareto front. The topology of the front is likely due to the metamodel, which may not have achieved full training convergence. Optimization achievements have been found significant for both \(R_T (-19 \div 18\%)\) and \(\nabla (+23 \div 28\%)\). Three optimal designs have been identified for (A) maximum total resistance reduction, (B) maximum displacement increase, and (C) minimum aggregate objective (with equal weights).

These designs perform better than the original at 18 kn, whereas the original design shows better performances in lower and higher speed ranges. This difference is mainly due to a significant change in length of the optimized designs, which reflects in a reduced Froude number at 18 kn. This motivates further investigations via robust design optimization for a stochastic speed ranges. Wave elevation and pressure distribution has been shown for design C, emphasizing the beneficial effects of the shape optimization.

The optimized designs finally show a significant increase of the wet surface (+20% or more) and variation of the water-plane area (from −11 to +1%). This motivates future extensions to Reynolds-averaged Navier-Stokes solvers (to address viscous effects), along with the proper
inclusion in the optimization of critical design constraints associated to the intact pitch and roll stability, seakeeping, and structural analysis, not considered at the current stage of the work.

ACKNOWLEDGEMENTS

Developments of the SBDO methodologies has been partially supported by the Office of Naval Research, NICOP grant N62909-15-1-2016, administered by Dr. Woei-Min Lin and, and by the Italian Flagship Project RITMARE, founded by the Italian Ministry of Education.

REFERENCES


OPTIMIZED DBD PLASMA ACTUATOR SYSTEM FOR THE SUPPRESSION OF FLOW SEPARATION OVER A NACA0012 PROFILE

LAURA PASQUALE*, DANilo DURANTE†, MATTEO DIEZ† AND RICCARDO BROGLIA†

* Aerospace Technology Centre, Faculty of Engineering
  University of Nottingham, UK
  e-mail: Laura.Pasquale@nottingham.ac.uk - Web page: http://www.nottingham.ac.uk

† National Research Council-Marine Technology Research Institute (CNR-INSEAN)
  Via di Vallerano 139, 00128 Rome, Italy
  e-mail: danilo.durante@cnr.it matteo.diez@cnr.it riccardo.broglia@cnr.it - Web page:
  http://www.insean.cnr.it

Key words: Flow separation control, Robust control, Plasma actuators, Multi-objective deterministic particle swarm optimization

Abstract. We address the problem of controlling the unsteady flow separation over an aerofoil, using plasma actuators. Despite the complexity of the dynamics of interest, we show how the problem of controlling flow separation can be formulated as a simple output regulation problem, so that a simple control strategy may be used. Different configurations are tested, in order to identify optimal positions of the actuator/sensor pairs along the aerofoil, as well as the corresponding references for the available real-time velocity measurements. A multi-objective deterministic particle swarm optimization algorithm is applied to identify the set of non-dominated configurations considering as objectives the time-averaged input signal and the drag-to-lift ratio. Accurate numerical simulations of incompressible flows around a NACA0012 profile at Reynolds \(Re = 20,000\) and angle of attack 15\(^\circ\) illustrate the effectiveness of the proposed approach, in the presence of complex nonlinear dynamics, which are neglected in the control design. Fast flow reattachment is achieved, along with both stabilisation and increase/reduction of the lift/drag, respectively. A major advantage of the presented method is that the chosen controlled outputs can be easily measured in realistic applications.

1 INTRODUCTION

Closed-loop flow control is aimed at altering a natural flow state into a more desirable state, which is chosen depending on control objectives. Crucial examples are: manipulation of flow separation, drag reduction, noise suppression, stall prevention \textit{etc}. Within this context, the incorporation of control theory into many open problems in fluid mechanics presents a host
of new opportunities, with a wide range of applications in disparate fields (e.g., gas turbines, aircraft, as well as ground and marine vehicles). The control input is usually an electric signal, which has to be converted to a physical quantity by means of an actuator. A new and original technology using non-thermal surface plasmas has witnessed a significant growth in interest in recent years [see, for instance, 6, 14], as they: have no moving parts; exhibit an extremely fast time-response; are characterised by low mass and low input power. These surface dielectric barrier discharge (DBD) actuators are used to accelerate the near-wall flow, thus modifying the velocity profile within the boundary layer.

In this paper, we focus on the robust feedback control of the flow separation using plasma actuators. In most flow control applications the objective is to suppress the separation bubble, as it is responsible for both a loss of the lift and an increase of the drag and it might lead to stall conditions. Recent works on feedback flow separation control include [1], where a slope-seeking algorithm is proposed to obtain maximum time-averaged lift. [4] proposed a retrospective cost adaptive algorithm to minimize the variation of the aerodynamic lift. However, the latter, which is the chosen output in both [1] and [4], cannot be measured in real-time in practical flow control applications.

Our objective is to solve the problem of directly controlling the unsteady flow separation using real-time velocity measurements, which are available in realistic applications [see, for instance, 16]. We propose this flow separation problem as a practical application of the new theoretical results in [11]. The aim of this paper is to show how, despite the high complexity of the system, a simple robust output regulator is sufficient to effectively suppress the flow separation along an aerofoil, using only one actuator/sensor pair. Accurate numerical simulations of flows past a NACA0012 profile are performed in order to test the control effectiveness, in the presence of complex nonlinear dynamics, which are neglected in the control design. A multi-objective deterministic particle swarm optimization (MODPSO) algorithm is finally applied to study the trade-off between the time-averaged input signal and the drag-to-lift ratio, varying the positions of the actuator/sensor pairs along the aerofoil, as well as the corresponding reference for the available real-time velocity measurements. Three sub-sets of non dominated configurations are identified and three solutions are selected accordingly for further investigations.

2 PROBLEM STATEMENT AND OBJECTIVES

This paper addresses the practical problem of robustly controlling the unsteady flow separation over an aerofoil, using the plasma actuator voltage as the control input and realistically available real-time velocity measurements as the control output. In particular, we aim to formulate and solve the flow separation problem, i.e., to make

$$\partial_n u_\tau(t,x)|_{\Gamma_N} = (\tau(x) \cdot \nabla u(t,x) \cdot n(x))|_{\Gamma_N} > 0, \quad (1)$$

as a simple output regulation problem, i.e., to make the measured output

$$y(t) = u_\tau(t,x_s) > \epsilon > 0. \quad (2)$$

Here: $u$ is the time-dependent flow velocity vector; $x$ and $x_s$ denote the spatial coordinates and the sensor location, respectively; $\Gamma_N$ represents the aerofoil boundary; $n$ and $\tau$ are the normal and tangent unit vectors to $\Gamma_N$, respectively.
Our objective is to design a robust output feedback control, along with a suitable reference signal \( y^* \) for \( y \), in order to suppress the flow separation along the aerofoil in unknown scenarios, depending on uncertain parameters, \( i.e., \) Reynolds number \( Re \) and angle of attack \( \beta \). To this end, we assume there exist suitable configurations of actuators and sensors, along with suitable references \( \varepsilon \) for the output \( y(t) \), which guarantee that, given a certain range for both \( Re \) and \( \beta \), the solution of the output regulation problem (2) implies the solution of the flow separation problem (1). This is formalised by the following assumption.

**Assumption 1.** For any \( \delta \geq 0 \) there exist some references \( \varepsilon > 0 \), \( a_{\varepsilon} > 0 \) and \( a_{\delta} \geq T_{\varepsilon} \), \( Re \in R_{Re} = [Re_m, Re_M] \), \( \beta \in R_\beta = [\beta_m, \beta_M] \).

We propose the application of the resulting robust output regulator to the incompressible Navier-Stokes equations. Several configurations are tested, thus allowing for the optimisation of the closed-loop performance. In particular, we aim to identify an optimal configuration \( \{\bar{x}_a, \bar{x}_s, y^*\} \), for which assumption 1 holds. Here: denotes the number of actuators; \( \bar{x}_a \) and \( \bar{x}_s \) denote the position of the actuator and sensor, with respect to the chord length, respectively.

### 3 FLOW MODEL

Let \( \Omega \) be an open bounded domain in \( \mathbb{R}^2 \) and let \( T > 0 \) denote the final time. The flow of an incompressible viscous Newtonian fluid can be described by the non-dimensionalised Navier-Stokes equations, which are derived from the conservation of mass and momentum, namely,

\[
\begin{align*}
\partial_t u & = -(u \cdot \nabla)u - \nabla p + \frac{1}{Re} \Delta u + f & \text{in } (0,T) \times \Omega, \\
0 & = \nabla \cdot u & \text{in } (0,T) \times \Omega,
\end{align*}
\]

with initial condition

\[ u(0, x) = u_0(x) \quad \text{in } \Omega, \]

and boundary conditions

\[
\begin{align*}
\mathbf{u}(t, x) & = \mathbf{g}(x) & \text{on } \Gamma_{in}, \\
\mathbf{u}(t, x) & = 0 & \text{on } \Gamma_0, \\
\left( \frac{1}{Re} \nabla \mathbf{u} - p \mathbf{I} \right) \mathbf{n} & = 0 & \text{on } \Gamma_{out}.
\end{align*}
\]

Here: \( x \in \Omega; \mathbf{n} \) denotes the unit outward normal vector on \( \partial \Omega = \Gamma_{in} \cup \Gamma_0 \cup \Gamma_{out}; \Gamma_{in}, \Gamma_{out} \) and \( \Gamma_0 \) denote the inflow, outflow and wall boundaries, respectively; \( \mathbf{u} : [0,T] \times \Omega \to \mathbb{R}^2 \) is the velocity vector; \( p : [0,T] \times \Omega \to \mathbb{R} \) is the pressure; \( I \in \mathbb{R}^{2 \times 2} \) is the identity matrix; \( Re = \rho U_\infty c / \mu \) is the Reynolds number; \( U_\infty \) is the free-stream velocity (in m/s); \( \rho \) is the fluid density (in kg/m\(^3\)); \( c = 0.1m \) is the chord length; \( f : [0,T] \times \Omega \to \mathbb{R}^2 \) is the total body force vector field, which depends the control inputs. The latter can be expressed as \( f(t, x) = c / \rho U_\infty^2 (f_x(t, x), f_y(t, x)) \) where \( f_x, f_y \) are the streamwise and normal component (in N/m\(^3\)). All the above listed functions are assumed to be sufficiently smooth. The wall-tangential velocity, evaluated at the selected sensor location \( x_s \),

\[ y(t) = u_\tau(t, x_s) = \tau(x_s) \cdot \mathbf{u}(t, x_s), \]

where

\[ \tau(x_s) = \nabla \times \mathbf{u}(t, x_s). \]
where $\tau$ denotes the tangent unit vector, is chosen as the measured output. Several models for the DBD actuator force have been proposed (see, for instance, [6] for a detailed review). Here, we select a modified version of the recent model proposed by [18], which demonstrated good agreement with the experimental data. The model is characterised by an exponential dependence on the spatial coordinates and, in particular, the force is modelled by a Rayleigh distribution [see 18]; thereby,

$$f(t, x) = f_\tau(t, x, n)\tau(x) + f_n(t, x, n)n(x) = I(t)\frac{\lambda f x_n}{(\sigma f)^2} e^{-x_n^2/(2\sigma f)^2} \tau(x), \quad (7)$$

where:

- $I(t) = k_v V(t) / V_m$ ($k_v \in \mathbb{R}, V_m = 1 \text{kV}$) is the total plasma force; $V(t) : \mathbb{R} \rightarrow \mathbb{R}$ is the amplitude variation of the operation voltage (in kV); $v(t) = V(t) / V_m$ is the corresponding non-dimensionalised voltage input, scaled by $V_m$; $f_\tau, f_n$ (in N/m$^3$) are the tangential and normal components, with respect to the aerofoil, of the force density, respectively; $x, y \geq 0$ are related to $x = (x, y)$ by a coordinate transformation and respectively refer to the tangent and normal components, relative to the geometry, in the reference frame centred in $x_a$. The parameters $\lambda_f = 1.6, \sigma_f = 1.9, k_v = 5200e^{1/2}\sigma_f / \lambda_f$, are chosen as in [18], where this model has been compared with particle image velocimetry (PIV) data, whilst, for sake of simplicity, a simple linear dependence of the body force on the applied peak-to-peak voltage is assumed here. System (3), (5), (4), (6), is discretised using χnavis, a general-purpose, second order, finite volume, multi-block, unsteady Reynolds averaged Navier-Stokes equations (uRaNSe) based solver, developed at CNR-INSEAN. For the sake of conciseness, details of the numerical solver are not given here, the interested reader is addressed to [2, 8].

### 4 FEEDBACK CONTROL PROBLEM

Consider an unknown stable linear system of the form:

$$\begin{cases}
\dot{\xi} = A\xi + Bv, \quad \xi(0) = \xi_0, \\
y = C\xi.
\end{cases} \quad (8)$$

System (8) might be seen as a linear approximation of the spatially-discretised Navier-Stokes equations (3), (5), (4), (6), which can be obtained, for instance, using spectral decomposition methods (see [15] for linear models of fluid systems based on the Koopman operator). Although system (8) cannot represent an accurate approximation of the actual nonlinear dynamics, we aim to show how a simple robust control algorithm based on an integral action, is sufficient to effectively suppress the separation bubble along the aerofoil using real-time velocity measurements at discrete locations.

Let $P(s) = C(sI - A)^{-1}B$, whose poles have all negative real part, be the open-loop transfer function of system (8). Denoting $\hat{\xi} = \xi - \xi^*$ and $\eta = -v^*$, where $\xi^*$ and $v^*$ denote the references for the state and control input, respectively, the error dynamics are given by

$$\begin{cases}
\dot{\hat{\xi}} = A\hat{\xi} + (v + \eta), \\
\dot{\eta} = 0, \quad \eta(0) = \eta_0, \\
\dot{\hat{y}} = C\hat{\xi},
\end{cases} \quad (9)$$

where $\tau$ denotes the tangent unit vector, is chosen as the measured output. Several models for the DBD actuator force have been proposed (see, for instance, [6] for a detailed review). Here, we select a modified version of the recent model proposed by [18], which demonstrated good agreement with the experimental data. The model is characterised by an exponential dependence on the spatial coordinates and, in particular, the force is modelled by a Rayleigh distribution [see 18]; thereby,

$$f(t, x) = f_\tau(t, x, n)\tau(x) + f_n(t, x, n)n(x) = I(t)\frac{\lambda f x_n}{(\sigma f)^2} e^{-x_n^2/(2\sigma f)^2} \tau(x), \quad (7)$$

where:

- $I(t) = k_v V(t) / V_m$ ($k_v \in \mathbb{R}, V_m = 1 \text{kV}$) is the total plasma force; $V(t) : \mathbb{R} \rightarrow \mathbb{R}$ is the amplitude variation of the operation voltage (in kV); $v(t) = V(t) / V_m$ is the corresponding non-dimensionalised voltage input, scaled by $V_m$; $f_\tau, f_n$ (in N/m$^3$) are the tangential and normal components, with respect to the aerofoil, of the force density, respectively; $x, y \geq 0$ are related to $x = (x, y)$ by a coordinate transformation and respectively refer to the tangent and normal components, relative to the geometry, in the reference frame centred in $x_a$. The parameters $\lambda_f = 1.6, \sigma_f = 1.9, k_v = 5200e^{1/2}\sigma_f / \lambda_f$, are chosen as in [18], where this model has been compared with particle image velocimetry (PIV) data, whilst, for sake of simplicity, a simple linear dependence of the body force on the applied peak-to-peak voltage is assumed here. System (3), (5), (4), (6), is discretised using χnavis, a general-purpose, second order, finite volume, multi-block, unsteady Reynolds averaged Navier-Stokes equations (uRaNSe) based solver, developed at CNR-INSEAN. For the sake of conciseness, details of the numerical solver are not given here, the interested reader is addressed to [2, 8].

### 4 FEEDBACK CONTROL PROBLEM

Consider an unknown stable linear system of the form:

$$\begin{cases}
\dot{\xi} = A\xi + Bv, \quad \xi(0) = \xi_0, \\
y = C\xi.
\end{cases} \quad (8)$$

System (8) might be seen as a linear approximation of the spatially-discretised Navier-Stokes equations (3), (5), (4), (6), which can be obtained, for instance, using spectral decomposition methods (see [15] for linear models of fluid systems based on the Koopman operator). Although system (8) cannot represent an accurate approximation of the actual nonlinear dynamics, we aim to show how a simple robust control algorithm based on an integral action, is sufficient to effectively suppress the separation bubble along the aerofoil using real-time velocity measurements at discrete locations.

Let $P(s) = C(sI - A)^{-1}B$, whose poles have all negative real part, be the open-loop transfer function of system (8). Denoting $\hat{\xi} = \xi - \xi^*$ and $\eta = -v^*$, where $\xi^*$ and $v^*$ denote the references for the state and control input, respectively, the error dynamics are given by

$$\begin{cases}
\dot{\hat{\xi}} = A\hat{\xi} + (v + \eta), \\
\dot{\eta} = 0, \quad \eta(0) = \eta_0, \\
\dot{\hat{y}} = C\hat{\xi},
\end{cases} \quad (9)$$
so that the control problem can be formulated as a disturbance rejection problem, where the reference input $v^* = -\eta$ can be viewed as a scalar disturbance, which matches the control input $v$ [see 11].

Similarly to [10], the control problem becomes to design a suitable feedback law $v(t)$ for system (8), based on the real-time measurement $y(t)$, in order to robustly regulate the latter to a given reference region (e.g. $y(t) \geq \epsilon > 0$). The key objective is to design $v$ such that the closed-loop trajectories of system (8) are guaranteed to evolve within some “safe” invariant set in different scenarios, depending on uncertain parameters (e.g., the Reynolds number $Re$ and angle of attack $\beta$). Therefore, on the basis of the recent results in [11], we design a robust output regulator guaranteeing exponential convergence of the regulation error: it only requires the system to have a non-zero steady-state gain of known sign.

4.1 Control algorithm

We translate the initial control objective (2) into the following: $y(t) \in \Omega_\epsilon = [\epsilon_m, \epsilon_M]$, where $\epsilon_m$ and $\epsilon_M$ are chosen positive constants. In particular, the lower bound for the output reference can be chosen in order to guarantee any a priori fixed requirement, such as, in the present application, the suppression of the separation bubble over the aerofoil; the upper bound can be chosen in order to limit the power consumption. Therefore, the control problem (similarly to [10]) becomes to design $v$ such that the chosen controlled output $y$ belongs to a “safe” compact set $\Omega_\epsilon = \Omega_{\epsilon_1} \times \Omega_{\epsilon_2} \times \ldots \times \Omega_{\epsilon_n}$. To this aim, the reference output $y^*$ is chosen as

$$y^*(t) = \begin{cases} \epsilon_m, & \text{if } y(t) < \epsilon_m, \\ y(t), & \text{if } y(t) \in \Omega_\epsilon, \\ \epsilon_M, & \text{if } y(t) > \epsilon_M. \end{cases}$$

(10)

The resulting control algorithm reads

$$\begin{cases} \dot{\hat{\eta}} = k \text{sign}(P(0))\dot{y}, & \dot{\eta}(0) = \hat{\eta}_0, \\ v = -\hat{\eta}. \end{cases}$$

(11)

The overall control algorithm (11), (10) depends on: the measured outputs $y$; the bounded references $y^*$; the known sign of $P(0)$; the positive design parameters $k$, $\epsilon_m$, $\epsilon_M$. Note that, when $\epsilon = \epsilon_m = \epsilon_M$ the control algorithm (11), (10) reduces to an output regulator with a constant output reference.

4.2 Stability Analysis

The stability properties of the closed-loop system are summarised by the following theorem.

**Theorem 1.** Consider the closed-loop system (8), (11), (10). Assume that $P(0) \neq 0$ with known sign. Then, for any initial condition $(\xi_0, \eta_0, \hat{\eta}_0)$, there exist sufficiently small $k^* > 0$, such that the regulation error $\tilde{y} = y(t) - y^*(t)$ and the control input error $v(t) - v^*(t)$ exponentially tend to zero, as $t$ tends to infinity, for any $0 < k \leq k^*$.

**Proof.** a). Case $\epsilon = \epsilon_m = \epsilon_M$. System (9) can be rewritten as

$$\dot{\tilde{Y}}(s) = P(s)(\nu(s) + \eta), \quad P(s) = \frac{n_P(s)}{d_P(s)}.$$

(12)
The stability of the closed-loop system is determined by the zeros of the transfer function

$$Q(s) = 1 + kP(s) \left(\frac{\text{sign}(P(0))}{s}\right) = \frac{n_Q(s)}{d_Q(s)}.$$  \hfill (13)

By the root locus, for sufficiently small $k > 0$, $r$ zeros of $Q(s)$ are sufficiently close to the $r$ poles of $P(s)$ and, therefore, they have negative real part. The remaining branch of the root locus starts from 0 in the $s$-plane with angle $\pi$, so that also the remaining zeros of $Q(s)$ have negative real part.

b). Case $\epsilon_m < \epsilon_M$. Let $\tilde{\eta} = v - v^* = \eta - \hat{\eta}$ and $\tilde{\chi} = [\hat{\xi}, \hat{\eta}]^T$. The closed-loop error dynamics can be written as

$$\dot{\tilde{\chi}} = \begin{bmatrix} A & B \\ -k \text{sign}(P(0))C & 0 \end{bmatrix} \tilde{\chi} = A_c \tilde{\chi},$$
$$\dot{\tilde{y}} = [C, 0]^T \tilde{\chi}.$$  

The characteristic polynomial of the closed-loop matrix $A_c$ can be computed as $p_{A_c}(s) = \det(sI_{r+1} - A_c) = sdp(s) + knP(s)\text{sign}(P(0)) = n_Q(s)$. Therefore, $A_c$ is Hurwitz, as its eigenvalues coincide with roots of $n_Q(s)$ and have negative real part for any sufficiently small $k$. Thus, there exist two symmetric, positive definite matrices $P$ and $Q$ satisfying the Lyapunov equation:

$$PA_c + A_c^TP = -Q.$$  

$V(t) = \tilde{\chi}^T(t)P\tilde{\chi}(t),$ satisfying

$$\alpha_1 \|\tilde{\chi}(t)\|^2 \leq V(t) \leq \alpha_2 \|\tilde{\chi}(t)\|^2,$$  \hfill (14)

where $\alpha_1, \alpha_2 > 0$ are positive constants. The time derivative of $V(t)$, along the trajectories of the closed-loop system satisfies the following inequality: $\dot{V} \leq -\tilde{\chi}^TQ\tilde{\chi} + 2\tilde{\chi}^TP\tilde{\chi} \leq -M\|\tilde{\chi}\|^2 \leq -\|Q\|\|\tilde{\chi}\|^2,$ where $M = \|Q\|$. Therefore, there exists an $\alpha_3 > 0$ such that

$$\dot{V} \leq -\alpha_3 \|\tilde{\chi}\|^2 \leq -\frac{\alpha_3}{\alpha_2} V,$$  \hfill (15)

thus implying the closed-loop boundedness and the exponential convergence to zero of both the regulation error $\tilde{y}(t)$ and the control input error $v(t) - v^*$, as $t$ tends to infinity. Let $\xi = \xi - \xi^*$ and $\hat{\eta} = v - v^*$. When the output vector belongs to the compact set $\Omega$, we have: $\xi \equiv 0$, $\hat{\xi} \equiv 0$, $\hat{\eta} \equiv 0$. Thus, for any $t \geq 0$ such that $\gamma(t) \in \Omega$, $\dot{V}(t) \equiv 0$. When the output does not belong to the reference region, there exist three positive constants $\alpha_1, \alpha_2, \alpha_3 > 0$ such that $V(t)$ and its time derivative satisfy (14) and (15), respectively. Therefore, for any $t \geq 0$ such that $\gamma(t) \notin \Omega$, $\dot{V}(t) < 0$ and the distance $d_P(\chi(t), \Omega_\chi) \equiv \inf_{\chi \in \Omega_\chi} \|\chi - \hat{\chi}\|_P \equiv \sqrt{\chi^T P \chi}$, between $\chi$ and its reference set $\Omega_\chi$ satisfies $\frac{d_P^2(\chi(t), \Omega_\chi)}{\alpha_2} \leq \alpha_3 \|\tilde{\chi}\|^2 \leq e^{-\alpha_1 \delta}$, where $\alpha = \alpha_3/\alpha_2$ and $\delta = V(0)/\alpha_2/\alpha_1$. Since $0 \leq V(t) \in \mathcal{C}_1$ is lower bounded and its derivative is semi-negative definite, it admits a finite limit [see 7, p. 61]. Closed-loop boundedness and exponential convergence of $\dot{V}(t)$ (and, therefore, of $\bar{\xi}$ and $\bar{\eta}$) to zero are thus guaranteed, according to Barbalats lemma, as $V(t)$ is uniformly continuous. Consequently, $\xi(t)$ converges to a constant reference $\bar{\xi} \in \Omega_\xi$ and $v(t)$ converges to a constant value $\bar{v}$, as $t$ tends to infinity. If $\bar{v} \notin \Omega_v$, then $\bar{\gamma} = C\bar{\bar{\xi}} = -P(0)\bar{\bar{v}} \notin \Omega_\xi$, which contradicts $\bar{\xi} \in \Omega_\xi$. Therefore, $\bar{\gamma} \in \Omega_v$ and the distance $d_P(\chi(t), \Omega_\chi)$ exponentially tends to zero, as $t$ tends to infinity.  \hfill $\square$
5 OPTIMIZATION METHOD

The MODPSO algorithm is used here for the minimization of the time-averaged input signal and the drag-to-lift ratio, $\dot{\phi} = \{\langle v \rangle, C_D/C_L \}^T$, versus the optimization variables, $\theta = \{\Delta_s, \epsilon_m \}^T \in \mathcal{D}$. Here: $\langle \cdot \rangle$ denotes the time-average; $\Delta_s = |\bar{x}_a - \bar{x}_s|$ is the distance between the sensor and the actuator, with respect to the chord length; $(\cdot)^T$ denotes the transpose of $(\cdot)$. The original PSO algorithm was introduced in [9], based on the social-behavior metaphor of a swarm of bees searching for food and belongs to the class of metaheuristic algorithms for single-objective derivative-free global optimization. Pinto et al. [13] proposed a multi-objective deterministic extension of the method as

$$
\begin{align*}
\mathbf{v}_i^{k+1} &= \gamma \left[ \mathbf{v}_i^k + c_1 (\mathbf{p}_i - \mathbf{\theta}_i^k) + c_2 (\mathbf{g}_i - \mathbf{\theta}_i^k) \right] \\
\mathbf{\theta}_i^{k+1} &= \mathbf{\theta}_i^k + \mathbf{v}_i^{k+1}
\end{align*}
$$

(16)

where $\mathbf{v}_i^k$ and $\mathbf{\theta}_i^k$ are the velocity and the position of the $i$-th particle at the $k$-th iteration, $\gamma$ is a constriction factor, $c_1$ and $c_2$ are the cognitive and social learning rate, and $\mathbf{p}_i$ and $\mathbf{g}_i$ are the cognitive and social attractor.

The algorithm formulation and setup is defined as suggested in [12]: the cognitive attractor $\mathbf{p}_i$ is the personal minimizer of the aggregate function $\Phi(\mathbf{\theta}_i) = \sum_{m=1}^{M} w_m \phi_m(\mathbf{\theta}_i)$, where $w_m = 1/M$ ($\forall m$) is the weight associated to the $m$-th objective function with $M$ the number of objective functions; the social attractor $\mathbf{g}_i$ is the closest point to the $i$-th particle of the Pareto front; the number of particles is set equal to 32, initialized over the domain $\mathcal{D}$ its boundary with a Hammersley distribution and non-null velocity [3]; the coefficients correspond to $\gamma = 0.721$, $c_1 = c_2 = 1.655$ [5]; a semi-elastic wall-type approach [17] is used to keep the particles within $\mathcal{D}$. The number of iterations is set to 1000.

6 RESULTS

Although the resulting robust control algorithm is designed on the basis of an unknown theoretical linear model, we propose its application to the flow separation control problem, along a NACA0012 profile, which is of practical interest. We only assume, coherently with assumption 1, a positive steady-state gain for any actuator/sensor pair.

6.1 Simulations

The computational grid has $N = 127,872$ total volumes and is divided into extremely fine actuator grids (see figure 1 right), a fine C-type inner grid (see figure 1 left) and coarser outer grids. The connections between the different grids are handled using an overlapping grid approach. The inner region around the profile has $320 \times 96$ volumes, in the tangent and normal direction, respectively; the points are clustered towards the wall, where the mesh spacing is equal to $2.1 \times 10^{-4}$. In the near wake region, $128 \times 192$ volumes, in the streamwise and vertical direction, respectively, are clustered around the wake of the profile.

The performance of the proposed control scheme (10), (11), is tested for the flow past a NACA0012 profile at $Re = 20,000$, in 21 different configurations: the actuator is placed at $\bar{x}_a = 0.02$, as preliminary tests (which are not reported here for the sake of brevity) showed a deterioration in the performance when it is moved further downstream; the distance $\Delta_a$ between
Figure 1: Computational grid around the NACA0012 profile (left) and actuator’s block (right).

Figure 2: Simulation results in the scenario $\epsilon_m = 0.05$.

Figure 3: Simulation results in the scenario $\epsilon_m = 0.1$.

The sensor and the actuator is varied between 0.3 and 0.8; three different lower bounds $\epsilon_m = 0.05$, $\epsilon_m = 0.1$, $\epsilon_m = 0.2$ of the reference set $\Omega$ are considered, while the upper bound is $\epsilon_M = \epsilon_m + 0.05$. The corresponding results are shown in figures 2-5. The controller is activated between $t_0 = 15$ and $t_f = T = 50$. The angle of attack is $\beta_0 = 15^\circ$. The output measurements $y(t) = u_{\tau}(t, x_s, y_s)$ are taken at $y_n = 0.0005$ above the aerofoil. The chosen control gain is $k = 20$. 

114
Figure 4: Simulation results in the scenario $\epsilon_m = 0.2$.

Figure 5: Time-averaged tangential velocity for $\epsilon_m = 0.005$ (top left), $\epsilon_m = 0.01$ (top right) and $\epsilon_m = 0.02$ (bottom).

6.2 Optimization

The Pareto front of the non-dominated solutions obtained by MODPSO is shown in Fig. 6 (a). The associated configurations in the $\Delta_s-\epsilon_m$ plane are shown in Figs. 6 (b) and (c) versus $\langle v \rangle$ and $C_D/C_L$, respectively. Three sub-sets are identified based on the clustering in the $\Delta_s-\epsilon_m$ plane. The clustering reflects clearly on the $\langle v \rangle$-$C_D/C_L$ trade-off. For each set, one solution is selected for further analysis. Specifically, solution 1 corresponds to $\Delta_s = 0.4$ and $\epsilon_m = 0.1$, providing a quite balanced compromise between $\langle v \rangle$ and $C_D/C_L$. Solution 2 has $\Delta_s = 0.3$ and $\epsilon_m = 0.2$ and one of the lowest values for $C_D/C_L$, whereas solution 3 corresponds to $\Delta_s = 0.2$ and $\epsilon_m = 0.1$ with a quite low value of $\langle v \rangle$.

The instantaneous vorticity contours for the selected solutions, are shown in figure 7: 101 non-dimensional vorticity levels over the range $[-15, 15]$, results for both with and without the actuation are reported for comparison purposes. Without the actuation, strong vortex structures are generated as a consequence of both the strong adverse pressure gradients and the boundary layer separation, which occurs on the upper side of the profile. The proposed control algorithm
significantly reduces the boundary layer separation and avoids the generation of large vortical structures.

![Graph](image)

Figure 6: Pareto front obtained by MODPSO (a), $\langle v \rangle$ (b), and $C_D/C_L$ (c) versus $\Delta_s$ and $\epsilon_m$ showing Pareto sub-sets and selected solutions

7 CONCLUSIONS

We addressed the practical problem of robustly controlling the unsteady flow separation over an aerofoil, using the plasma actuator’s voltage as the control input and realistically available real-time velocity measurements as the control output. In particular, we formulated the flow separation problem as a simple output regulation problem and solved the latter by designing a robust feedback control algorithm. Accurate numerical simulations of flows past a NACA0012 at Reynolds $Re = 20,000$ and angle of attack $\beta = 15^\circ$ are performed in order to both test the control effectiveness and optimize the performance of the closed-loop system. Although the proposed controller is simple, as it is based on an integral action, it is able to effectively suppress the separation bubble, as well as the shedding vortices. Different configurations were tested, to the aim of identifying optimal positions of the actuator/sensor pairs along the aerofoil and the corresponding references for the available real-time velocity measurements. Finally, a multi-
objective deterministic particle swarm optimization algorithm was applied to identify the Pareto set of non dominated configurations considering as objectives the time-averaged input signal and the drag-to-lift ratio. Three sub-sets of non dominated configurations were identified based on the solution clustering and, for each set, one solution was selected for further investigations and demonstration of the methodology.

8 ACKNOWLEDGEMENTS

The research leading to these results has received fundings from: the People Programme (Marie Curie Actions) of the European Unions Seventh Framework Programme (FP7/2007-2013) under REA grant agreement no 608322; the present work was also supported by the Project RESMARE (Ricerca E Servizi per il MARE) funded by the Lazio Region.

References


TOWARDS THE HIGH-FIDELITY MULTIDISCIPLINARY DESIGN OPTIMIZATION OF A 3D COMPOSITE MATERIAL HYDROFOIL

SILVIA VOLPI*, MATTEO DIEZ†** AND FREDERICK STERN*

*IIHR-Hydroscience and Engineering, The University of Iowa
100 C. Maxwell Stanley Hydraulics Laboratory, Iowa City, IA 52242, USA
e-mail: frederick-stern@uiowa.edu

†CNR-INSEAN, National Research Council-Marine Technology Research Institute
Via di Vallerano 139, 00128 Rome, Italy

Key words: Multidisciplinary design optimization, Fluid-structure interaction, Design space dimensionality reduction, Surrogate models, Adaptive sampling

Abstract. The development of a multidisciplinary design optimization (MDO) architecture for high-fidelity fluid-structure interaction (FSI) problems is presented with preliminary application to a NACA 0009 3D hydrofoil in metal and carbon-fiber reinforced plastic materials. The MDO methodology and FSI benchmark solution are presented and discussed. The computational cost of the MDO is reduced by performing a design space dimensionality reduction beforehand and integrating into the architecture a variable level of coupling between disciplines, a surrogate model, and an adaptive sampling technique. The optimization is performed using a heuristic global derivative-free algorithm. The MDO method is demonstrated by application to an analytical test problem. Current stage of research includes preliminary test problem optimization, validation of the hydrofoil FSI against experimental data, and design space assessment and dimensionality reduction for the hydrofoil model.

1 INTRODUCTION

The design of water-borne vehicles and components relies on high-fidelity simulations, organized to provide the performance of alternative designs in a variety of operating and environmental conditions. The simulation-based design (SBD) approach generally includes a hydrodynamic and/or structural solver, a design modification tool, and optimization algorithms and has been successfully applied to deterministic and stochastic optimization of a variety of ships. Lately, [2] successfully applied SBD methods based on RANS to the deterministic optimization of a fast catamaran in calm water. Extensions to stochastic SBD for ships in realistic ocean conditions were presented in [3] and applied to the same catamaran model. Both deterministic and stochastic optimization methods relied solely on a high-fidelity hydrodynamic solver and did not account for the elastic structural response of the ship.

Fluid-structure interaction (FSI) may be of paramount significance in specific phenomena, such as slamming, springing, and whipping of ships, as well as hydro-elastic effects of rudders, propellers, and appendages in general. To predict accurately the effects of FSI on the hydro-structural performance in SBD, high-fidelity FSI solvers are required along with their validation in complex realistic conditions. Recent research by the authors showed the full-scale validation of a partitioned FSI solver based on one- and two-way URANS/FE coupling for a high-speed craft with composite panels, slamming in waves [16]. The effects of FSI were found significant,
motivating further FSI methodology development and the integration of FSI into the SBD via multidisciplinary design optimization (MDO).

MDO refers to optimization procedures where the design performance depends on several interconnected disciplines. MDO architectures define the coupling between disciplines (such as strong or weak) and the sequence of tasks required to achieve both the multidisciplinary consistency and the solution of the design optimization problem. In the context of FSI, the availability of experimental data is essential to validate the multidisciplinary consistency achieved by the process. Lately, FSI data [17] of a NACA 0009 3D hydrofoil have been used as a benchmark for numerical hydrodynamic [8], FSI and MDO studies, investigating the effects of multiple materials (metals and composite).

High-fidelity MDO of complex engineering problems represents a technological challenge. Complexity of the multidisciplinary analysis and an often-large number of design variables are critical factors affecting the computational cost. Design space dimensionality reduction techniques and surrogate models can be used to alleviate the computational resource consumption. Design space dimensionality reduction is generally performed by sensitivity analysis, requiring fully-coupled multidisciplinary analyses and not addressing the interdependence of design variables. To overcome these limitations, an offline design space dimensionality reduction method for shape optimization has been recently presented [4],[5] based on the Karhunen-Loève expansion (KLE) of the shape modification vector. The technique adopts a purely geometrical perspective and precedes any analysis/optimization process. Surrogate models have been extensively used in several engineering fields. Examples of multidisciplinary applications are given in [13] and [14]. The combination of surrogate models and adaptive sampling techniques has been investigated in earlier research [18],[15] showing promising improvements in accuracy and efficiency.

The objective of the current research is the development of an MDO architecture for high-fidelity optimization of complex FSI problems, with application to a NACA 0009 3D hydrofoil in carbon fiber-reinforced plastic (CFRP). The MDO methodology and FSI benchmark solution are presented and discussed.

Specifically, the design space dimensionality reduction is performed beforehand by extension of the combined distributed/concentrated parameters KLE presented in [6] to purely geometrical quantities. The identification of the optimal design is achieved by sequential surrogate-based global derivative-free optimization using dynamic radial basis functions (DRBF) [15] and deterministic particle swarm optimization (DPSO) [1]. The multi-criterion adaptive sampling (MCAS) [7] is extended to multi-disciplinary optimization allowing for a variable level of coupling between fluid and structural dynamics. As the analysis advances, the design space is explored and the multidisciplinary consistency refined. A steady two-way coupled FSI is solved using Gauss-Seidel iterations. Current status of the research includes: (a) a preliminary analytical test problem optimization (including comparison of the current architecture to a standard multidisciplinary feasible, MDF [10], approach), (b) the validation of FSI analysis against experimental data for the hydrofoil in stainless steel, aluminum, and CFRP, and (c) design space assessment and dimensionality reduction.

2 PROBLEM STATEMENT AND NUMERICAL METHODS

2.1 Multidisciplinary design optimization: statement of the problem and architecture

The multidisciplinary design optimization problem for FSI is formulated as follows

\[
\min_{\mathbf{u}} f(\mathbf{u}, \mathbf{y}(\mathbf{u}, \mathbf{y}))
\]
subject to \( c_0[u, y(u, y)] \leq 0 \)
\( c_F[u_0, u_F, y_F(u_0, u_F, y_F)] \leq 0 \)
\( c_S[u_0, u_S, y_S(u_0, u_S, y_S)] \leq 0 \)

where \( f \) is the objective function, \( c \) is the set of constraint functions, \( u \) is the set of design variables, and \( y \) is the set of coupling variables from fluid and structural analyses. Design variable bounds are handled directly by the optimizer. The subscript \((\cdot)_F\) indicates the fluid analysis, \((\cdot)_S\) refers to the structural analysis, and \((\cdot)_0\) indicates that the variable/function is shared by both disciplines. The inequality constraints \( c \) are handled by a linear penalty function. This allows for recasting the formulation in Eq. 1 into the following unconstrained optimization

\[
\min_u f_p[u, y(u, y)]
\]  

(2)

where \( f_p \) is the penalized objective function defined as

\[
f_p = f + \gamma \sum_{i=1}^{N_{c_F}} \max \{c_{0_F}, 0\} + \gamma \sum_{i=1}^{N_{c_S}} \max \{c_{0_S}, 0\} + \gamma \sum_{i=1}^{N_{c_F}} \max \{c_{0_F}, 0\} 
\]  

(3)

and \( \gamma \) is a penalty coefficient.

The optimization is solved using sequential surrogate models trained by objective function values provided by the FSI analysis. The density of the training points is improved by infill of new samples at each iteration. The FSI coupling is initially loose and gets tighter as the optimization advances. The procedure is shown in the block diagram of Figure 1.

Figure 1: MDO solution procedure and FSI loop

The DRBF surrogate model provides the prediction of the function \( \hat{f}(u) \) as the expected value of a sample of standard RBF predictions over a stochastic distribution of a tuning parameter associated with the RBF kernel. The sampling uncertainty \( U_S(u) \) is the 95% confidence interval of the stochastic sample. Details of the method can be found in [15]. The FSI loop is driven by Gauss-Seidl iterations between the fluid and the structural solvers. A coupling uncertainty \( U_C \) is defined based on the convergence of the Gauss-Seidl loop and quantified by the difference between two consecutive iterations.

The adaptive sampling technique MCAS identifies groups of new samples for training the DRBF model aiming at balancing the surrogate model accuracy and the search for the global
minimizer. In case of single-discipline optimization, this is performed by solving the multi-objective optimization problem

$$\min_u f_p(u) \text{ and } \max_u U(u)$$

(4)

where $U$ is the uncertainty of the surrogate model $U_s$, depicted in Figure 2a. The resulting Pareto front is down-sampled to identify $I$ equally spaced points along the curve as shown in Figure 2b. Details of the method are provided in [7].

![Figure 2](image)

**Figure 2**: Single-discipline (a, b) and multidisciplinary (c, d) MCAS

In case of multidisciplinary optimization, MCAS accounts for both $U_s(u)$ and $U_c(u)$. As shown in Figure 2c, $U_s$ is continuous and goes to zero at the training points; $U_c$ is instead discrete and defined only at the training points. Two independent Pareto sets, $\varphi_S$ and $\varphi_C$, are built using $U = U_s$ and $U = U_c$ in Eq. 4 and are depicted in Figure 2d. A new set $\varphi$ is defined taking the non-dominated solutions of $\varphi_S \cup \varphi_c$, with the additional constraints, $U > U_{\text{min}}$ (in order to avoid overshooting in training the surrogate model) and $\tilde{f} - \tilde{f}^* > U_{\text{tot}}$ where $U_{\text{tot}} = \sqrt{U_s^2 + U_c^2}$ (‘*’ indicates current optimum values). $\varphi$ is down-sampled as in the standard MCAS. If a sample is determined that was originally in $\varphi_c$ the corresponding function value in the training set is updated by refining the FSI analysis. If a sample is determined that was originally in $\varphi_s$ a new point is added to the training set. The method is referred to as MCAS with concurrent uncertainties (MCAS-CU). The multi-objective version [11] of the DPSO algorithm is applied to solve Eq. 4 and extract the Pareto sets.

### 2.2 Fluid-structure interaction

The fluid is modeled by the continuity and RANS equations for incompressible Newtonian viscous flow. The CFD finite difference code CFDShip-Iowa V4.5 [9] is used to solve continuity and momentum equations along with the turbulence equations for the Menter’s blended $k$-$\omega/k$-$\varepsilon$ model. The structural equation of elastic motion is solved by a modal expansion, where modes and frequencies are provided by the computational structural dynamics (CSD) solver ANSYS Mechanical APDL V15, a commercial FE code.

In the Gauss-Seidl iterations for steady FSI, $y_F$ and $y_S$ are the hydrodynamic load predicted by the fluid solver and the displacement predicted by the structural solver, respectively, at the fluid-structure interface. The hydrodynamic load is transferred from the fluid to the structure mesh by Gauss interpolation of the force distribution $f(x)$ so that $f_F(x) \approx f_S(x)$ and $F_F \approx F_S$, where $F = \int f(x)dx$ is the total force. The current interpolation approach allows for force conservation, while moments are conserved only in an asymptotic sense, i.e. for a mesh size that goes to zero. When transferring the displacement from the structure to the fluid mesh, the deformation is interpolated in the entire volume representing the fluid domain. The fluid volume mesh is structured with indices $I, J,$ and $K$, where $J = 1$ corresponds to the fluid-solid interface.
First, the \( J = 1 \) surface is deformed by Gauss interpolation so that \( \delta_S(x) \approx \delta_p(x) \). Then, the volume inner nodes are displaced by linear interpolation between interface \((J = 1)\) and outer \((J = J_{\text{MAX}})\) boundary layer surfaces. Details and applications of the FSI routine are given in [16].

### 2.3 Shape modification, design-space assessment and dimensionality reduction

The shape modification is performed using free-form deformation (FFD) [12]. The method allows for smooth deformations of an arbitrary object by deformation of the 3D space embedding the object. The deformation is propagated in the space from displacement of discrete control points. The design variables are the displacements of the control points.

The design space assessment and dimensionality reduction are performed considering the breakdown of the geometric variance associated to the shape modification space. The method is based on the KLE of the shape modification vector [4]. It is extended here to a combined distributed/concentrated geometrical parameter vector, similarly to what introduced in [6] to integrate physical parameters in the design space assessment.

The aim of the KLE is to find an optimal basis of orthonormal functions for the linear representation of the shape modification \( \hat{\mathbf{y}}(\mathbf{x}, \mathbf{u}) = \sum_{k=1}^{N^G} \alpha_k(\mathbf{u}) \psi_k(\mathbf{x}) \), where

\[
\hat{\mathbf{y}}(\mathbf{x}, \mathbf{u}) = \begin{cases}
\hat{\mathbf{y}}(\mathbf{x}, \mathbf{u}), & \mathbf{x} \in \mathcal{D} \\
\hat{\theta}(\mathbf{x}, \mathbf{u}), & \mathbf{x} \in \mathcal{C}
\end{cases}
\quad \text{and} \quad \psi_k(\mathbf{x}) = \begin{cases}
\varphi_k(\mathbf{x}), & \mathbf{x} \in \mathcal{D} \\
\vartheta_k(\mathbf{x}), & \mathbf{x} \in \mathcal{C}
\end{cases} \tag{5}
\]

where \( \mathcal{D} \) and \( \mathcal{C} \) are the domains of the distributed and concentrated modifications \( \hat{\mathbf{y}} \) and \( \hat{\theta} \), respectively. In the present work, \( \hat{\mathbf{y}} \) is the distributed shape modification vector (displacement) and \( \hat{\mathbf{\theta}} \) includes twist and camber at specified sections. The basis retaining the maximum variance is determined by solution of an eigenvalue problem [4]. The KLE eigenvalues \( \lambda_k \) represent the design variability (variance) associated to the corresponding KLE modes \( \psi_k \). Provided that \( \lambda_k \geq \lambda_{k+1} \), the reduced-dimensionality design space is built by truncating the linear expansion to the order \( N \), so as to resolve a desired level of geometric variability \( \sum_{k=1}^{N} \lambda_k \).

The coefficients \( \alpha_k \) are used as new design variables of the reduced-dimensionality design space.

### 3 ANALYTICAL TEST CASE

The MDO architecture is applied to the following two-dimensional test problem:

\[
\begin{aligned}
\min_{\mathbf{u}} \quad & f(\mathbf{u}) = u_1^2 + u_2 + y_1 + e^{-y_2} \\
\text{with} \quad & y_1(\mathbf{u}, y_2) = 100 + u_1 + u_2 - 0.2y_2 \\
& y_2(\mathbf{u}, y_1) = \sqrt{|y_1|} + 10 + u_2 \tag{6}
\end{aligned}
\]

Box-constraints are \(-10 \leq u_1 \leq 25 \) and \(-25 \leq u_2 \leq 10 \). The problem is solved using a convergence criterion based on the coupling uncertainty \( U_c \) in the neighborhood of the optimum. \( U_c \) and \( U_{\text{min}} \) are set equal to \( 10^{-4} \). The RBF kernel used is a power law with exponent uniformly distributed between 1 and 3. Results are compared with the standard MDF architecture. The latter applies single-objective DPSO directly to objective function evaluations reaching multidisciplinary consistency (with tolerance \( 10^{-4} \)). Both methods are initialized by a Hammersley sequence distribution of 16 points. Single- and multi-objective DPSO uses 16 particles for 100 iterations.

The convergence of the optimum is depicted in Figure 3a including the optimal solution as predicted by the surrogate model \( f_s \), the total uncertainty in prediction \( U_{\text{tot}} \), and the true value of \( f \). In approximately 4 iterations \( U_{\text{tot}} \) reduces significantly (0.01\%) and the surrogate model
prediction appears accurate. A breakdown of the uncertainty is provided in Figure 3b showing $U_s$ and $U_c$. The current architecture outperforms a standard MDF with equivalent tolerance for $U_c$, as shown by Figure 3c. The number of function evaluations needed to achieve the optimum is smaller by an order of magnitude. The distribution of samples for the two architectures is shown in Figure 4.

![Figure 3](image1)

**Figure 3**: Convergence of (a) optimum, (b) uncertainties, and (c) comparison with MDF

![Figure 4](image2)

**Figure 4**: Current architecture sampling

### 4 INDUSTRIAL TEST CASE

The 3D hydrofoil (Figure 5 and Table 1) is tapered with streamlined NACA 0009 cross-sections and is clamped at the root section. An experimental study was carried out at the University of Tasmania-AMC to investigate the steady hydro-elastic behavior of the hydrofoil comparing different materials, including stainless steel, aluminum, and two sandwich structures manufactured with CFRP external skins and inner foam core [17]. The CFRP hydrofoils differ in fiber orientation: CFRP00 fibers are aligned with the span whereas CFRP30 fibers are at a 30 degrees angle. The models have thicker trailing edge than standard NACA 0009 to accommodate the composite material. Lift $C_L$, drag $C_D$, and pitching moment $C_M$ coefficients, tip displacement $\delta$ and twist $\theta$ are available from water tunnel testing for several Reynolds number and angles of attack.
4.1 Fluid-structure interaction

Steady two-way coupling FSI is performed at an angle of attack \( \alpha \) equal to 8 degrees and a Reynolds number equal to 0.6x10^6. The condition is in the pre-stall range providing steady hydrodynamics. The two-way FSI analysis focuses on the validation of \( C_L, C_D, C_M, \delta \), and \( \theta \).

Figure 6 show the \( C_L, C_D \), and \( C_M \) curves including CFD simulations, FSI simulations, and experimental data (EFD/ESD). The deformation of steel and aluminum hydrofoils is small, affecting only slightly the hydrodynamic forces. Both CFD simulations for rigid body and FSI simulations provide a good approximation of the curves. At \( \alpha = 8 \) degrees, the errors for \( C_L, C_D \), and \( C_M \) between FSI and experiments (as percentage of the experimental dynamic range) are -1.11, 0.97, and -0.72% for steel and -1.43, 2.01, and -0.11% for aluminum. The deformation of the CFRP00 hydrofoil is larger than the metals and for the CFRP30 is particularly significant, leading to a visible discrepancy between experimental and CFD curves (with errors up to 10%). The errors between FSI and experiments are smaller and equal 0.46, 1.03, and 1.11% for CFRP00 and 1.77, 2.00, and 1.05% for CFRP30 for \( C_L, C_D \), and \( C_M \), respectively. Overall, the FSI prediction of hydrodynamic forces is found accurate with a maximum error close to 2%.

The hydrodynamic efficiency \( C_L/C_D \) is shown in Figure 7. The experimental values for steel, aluminum, CFRP00, and CFRP30 are 20.1, 21.4, 19.3, and 21.3. Those computed by FSI are 18.7, 18.7, 18.3, and 19.1. The experimental data for aluminum shows large peaks between 4 and 8 degrees, which appear likely to be outliers. Excluding those, CFRP30 provides the largest efficiency overall.

Figure 8 shows the hydrofoil shape computed by the FSI focusing on the tip deformation. The tip displacement for steel and aluminum is small (less than 3% of span) and the twist is

**Table 1: 3D hydrofoil parameters**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chord length at root</td>
<td>0.12 m</td>
</tr>
<tr>
<td>Chord length at tip</td>
<td>0.06 m</td>
</tr>
<tr>
<td>Mean chord length</td>
<td>0.09 m</td>
</tr>
<tr>
<td>Span length</td>
<td>0.3 m</td>
</tr>
<tr>
<td>Aspect ratio</td>
<td>3.33</td>
</tr>
<tr>
<td>Cross-section</td>
<td>NACA0009</td>
</tr>
</tbody>
</table>
negligible. The error between FSI and experiments for steel and aluminum is -7.6 and -18% of the experimental pre-stall range. The tip displacement for CFRP00 is approximately 3% of the span whereas it is 6% of span for CFRP30. 0.5 degrees positive tip twist is found for CFRP00 and 1.5 degrees of negative twist for CFRP30. The errors for CFRP00 and CFRP30 are -6.2 and -13% for $\delta$ and 36% and 1.2% for $\theta$, respectively. Overall, $\delta$ is under-predicted by the FSI but the trend is captured.

[Figure 7: Hydrodynamic efficiency]

[Figure 8: Deformation and contour of the $x$-velocity at 95% of the span]

CFRP30 results for $\theta$ are consistent with the experimental hydrodynamic curves, showing a reduction of forces and moment compared to the CFD. The stall-delay effect is due to the negative twist of the tip section, which reduces the effective angle of attack. The associated efficiency increase is due to the reduction of tip vortex intensity. Figure 9 depicts the non-dimensional velocity distributions through the vortex core comparing CFRP30 with the rigid hydrofoil. The CFRP30 shows a smaller fluctuation in the predominant velocity component. Moreover, at the tip vortex core and downstream the tip trailing edge ($x = 0.06m$), the second invariant of the rate of strain tensor $Q$, taken as criterion for vortex identification, is equal to 11600 for rigid body and 5500 for CFRP30. Overall, the FSI simulation captures the behavior of the structure giving a realistic description of the physics involved.
4.2 Shape modification and design space assessment

Four distributions of FFD control points are compared, as shown in Figure 10. Control points are allowed to move in the chord- (x) and thickness-wise (y) directions, whereas their position along the span (z) is fixed. Table 2 summarizes the FFD setups including the number of associated design variables. Additional design variables are included in the geometry modification. Rigid displacement and rotation of the sections are imposed. Bounds for all x/y displacements are set equal to ±20% of the mean chord; bounds for the rotations are set equal to ±15 degrees. Overall, five design spaces are analyzed and summarized in Table 3. DS1 to 4 have no rigid section displacement and uses a linear distribution for section rotation. DS5 includes independent rigid displacements and rotations at 5 sections.

![Figure 10: FFD control points distribution](image)

**Table 2: FFD setups**

<table>
<thead>
<tr>
<th>Setup</th>
<th>Control points distribution</th>
<th>Number of variables</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2x2x2</td>
<td>16</td>
</tr>
<tr>
<td>2</td>
<td>5x2x5</td>
<td>100</td>
</tr>
<tr>
<td>3</td>
<td>10x2x10</td>
<td>400</td>
</tr>
<tr>
<td>4</td>
<td>20x2x20</td>
<td>1600</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Design space</th>
<th>FFD setup</th>
<th>Additional DOF</th>
<th>Total number of design variables</th>
</tr>
</thead>
<tbody>
<tr>
<td>DS1</td>
<td>1</td>
<td>1</td>
<td>17</td>
</tr>
<tr>
<td>DS2</td>
<td>2</td>
<td>1</td>
<td>101</td>
</tr>
<tr>
<td>DS3</td>
<td>3</td>
<td>1</td>
<td>401</td>
</tr>
<tr>
<td>DS4</td>
<td>4</td>
<td>1</td>
<td>1601</td>
</tr>
<tr>
<td>DS5</td>
<td>2</td>
<td>12</td>
<td>112</td>
</tr>
</tbody>
</table>

Table 4 gives the geometric variance $\sigma^2$ of the five design spaces computed using a standard KLE method. The comparison of DS1 to 4 shows that increasing the number of control points in the FFD reduces $\sigma^2$. The comparison of DS2 and DS5, which share the same FFD setup, shows that using additional rigid displacements and rotations increases $\sigma^2$. Overall, DS5 has the largest $\sigma^2$. Figure 11 displays the KLE eigenvalues convergence. The number of eigenvalues required to retain the 50, 75, 90, 95, 99, and 99.9% of $\sigma^2$ is summarized in Table 4 and labeled in Figure 11.

![Figure 11: Eigenvalues convergence](image)

**Table 4: Design space variance assessment**

<table>
<thead>
<tr>
<th>Design space</th>
<th>$\sigma^2$</th>
<th>No. of modes for X% of $\sigma^2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>DS1</td>
<td>$4.34x10^{-9}$</td>
<td>3 7 10 12 14 16</td>
</tr>
<tr>
<td>DS2</td>
<td>$1.42x10^{-9}$</td>
<td>8 16 27 36 54 77</td>
</tr>
<tr>
<td>DS3</td>
<td>$6.90x10^{-10}$</td>
<td>16 32 55 73 116 175</td>
</tr>
<tr>
<td>DS4</td>
<td>$3.50x10^{-10}$</td>
<td>27 59 105 141 226 347</td>
</tr>
<tr>
<td>DS5</td>
<td>$4.61x10^{-9}$</td>
<td>3 5 11 19 33 63</td>
</tr>
</tbody>
</table>
Table 5: Combined distributed/concentrated geometric parameters for KLE analysis

<table>
<thead>
<tr>
<th>Displacement weight</th>
<th>Twist weight</th>
<th>Camber weight</th>
<th>No. of modes for $\chi^2$ of $\sigma^2$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>50</td>
</tr>
<tr>
<td>1.0</td>
<td>0.0</td>
<td>0.0</td>
<td>3</td>
</tr>
<tr>
<td>0.9</td>
<td>0.05</td>
<td>0.05</td>
<td>3</td>
</tr>
<tr>
<td>0.8</td>
<td>0.1</td>
<td>0.1</td>
<td>4</td>
</tr>
<tr>
<td>0.6</td>
<td>0.2</td>
<td>0.2</td>
<td>4</td>
</tr>
<tr>
<td>0.5</td>
<td>0.25</td>
<td>0.25</td>
<td>3</td>
</tr>
<tr>
<td>0.333</td>
<td>0.333</td>
<td>0.333</td>
<td>2</td>
</tr>
</tbody>
</table>

When performing combined distributed/concentrated parameter KLE, a weight is assigned to distributed (displacements) and concentrated (twist and camber) modifications. DS5 is evaluated comparing six sets of relative weights. The number of eigenvalues required to retain the 50, 75, 90, 95, 99, and 99.9% of $\sigma^2$ is summarized in Table 5. Using even weights for the three parameters provides the most efficient dimensionality reduction.

(a) Distributed parameter KLE  
(b) Combined distributed/concentrated parameter KLE

Figure 12: Target geometry representation

Figure 13: Eigenvectors of DS5 representation with combined distributed/concentrated parameter KLE

A nearly optimal shape inspired by [8] is used as a target to test the accuracy of the reduced-dimensionality representation of the shape modification. Figure 12 shows the geometry reconstruction using a finite number of KLE modes comparing standard (a) and combined distributed/concentrated parameter KLE (b). The design space representation through the latter
method outperforms standard KLE in representing the target geometry. The first three eigenvectors used by the combined distributed/concentrated parameter KLE for representing DS5 are depicted in Figure 13.

4.3 Multidisciplinary design optimization

Ongoing MDO studies aim at the design optimization of the hydrofoil for maximizing the hydrodynamic efficiency at specified conditions. The multiple design choices offered by the CFRP material are investigated and a combined shape and structural design optimization is sought after.

9 CONCLUSIONS AND FUTURE RESEARCH

The development of an MDO architecture for high-fidelity FSI problems has been shown with preliminary application to a NACA 0009 3D hydrofoil in metal and CFRP materials. The MDO methodology and FSI benchmark solution were presented and discussed.

The methodology for high-fidelity MDO integrates design space dimensionality reduction, surrogate modeling, and adaptive sampling into a global derivative-free optimization framework. The architecture performs sequential surrogate-based optimization refining the accuracy of the analysis by infill of new samples and improvement of the multidisciplinary consistency. The MDO methodology was applied to a two-dimensional analytical test problem and compared with a standard MDF architecture employing the same optimization algorithm and convergence criterion. The current method outperforms the MDF requiring fewer function evaluations by an order of magnitude.

The FSI of the hydrofoil original geometry was studied and compared with available experimental data. The simulations showed an accurate description of the physics and prediction of lift and drag forces, pitching moment, hydrodynamic efficiency, tip displacement and twist. Results were shown for stainless steel, aluminum, and two types of carbon fiber-reinforced plastic (CFRP) materials. The negative tip twist provided by one of the CFRP materials correlates with lower tip vortex intensity, stall delay, and higher hydrodynamic efficiency.

Several design spaces for the hydrofoil were assessed by Karhunen–Loève expansion (KLE), showing geometric variability, convergence of the KLE eigenvalues, and capability in representing a target nearly optimal geometry. A novel KLE formulation based on combined distributed/concentrated geometrical parameters was used to build a more efficient reduced-order representation of the design space than standard KLE.

Future research includes the sensitivity analysis along KLE-provided design variables. The hydrodynamic shape optimization of the rigid hydrofoil will be performed along with the MDO of the aluminum and CFRP models.

ACKNOWLEDGEMENTS

The research is supported by the Office of Naval Research, grant N00014-14-1-0195 and NICOP grant N62909-15-1-2016, administered by Dr. Thomas Fu, Dr. Ki-Han Kim, and Dr. Woei-Min Lin. Matteo Diez is also grateful to the National Research Council of Italy for its support through the Short-Term Mobility Program 2015-2016.

REFERENCES


WAVE RESISTANCE MINIMISATION IN PRACTICAL SHIP DESIGN

Hoyte C. Raven, Thomas P. Scholcz

Maritime Research Institute Netherlands
P.O.Box 28, 6700AA Wageningen, Netherlands
e-mail: h.c.raven@marin.nl

Keywords: Optimisation, wave resistance, surrogate-based, multi-objective

Summary  A practical and efficient system is described for ship hull form optimisation and its application for minimising wave resistance. Parametric hull form deformations are defined in a CAD system, specific for the case considered and related with flow aspects to be addressed. Surrogate-based global optimisation is applied for multi-objective problems, such as optimisation for a ship’s operational profile.

INTRODUCTION

The wave resistance of a ship at speed in still water is generally known to be most sensitive to hull form variations. Consequently, while not being the largest resistance component for most vessels, it may offer good possibilities for reducing the ship’s fuel consumption and is, therefore, always considered in hydrodynamic hull form design. Free-surface potential-flow codes can provide a fast and accurate evaluation for most of the ship wave pattern. Available insight in the physics often permits to reduce the wave resistance by hull form modifications in a stepwise procedure [1,2]. At MARIN, since many years this is frequently supported by analysis of design trends using systematic hull form variations [3].

For fine-tuning, for simultaneous variation of a larger number of degrees of freedom, or for finding an optimal tradeoff between several design points, automatic optimisation offers most useful additional possibilities. Therefore we apply it more and more frequently in practice. The present paper describes the setup of the wave resistance minimisation system that we apply in practical ship hull form design projects. Several hull form optimisation methods have been proposed, e.g. [4,5,6,7]. However, our method has some particular features, having been developed based on the typical use at MARIN:

- In most cases, our work addresses the final hull form design in a later stage of the design process. Main dimensions, main coefficients, LCB and an initial hull form have already been fixed, and several ‘hard points’ may need to be respected. Therefore, we must be able to make detailed hull form improvements, confined to the design aspects and parts of the hull that may be modified. Flexibility of the parametrisation is therefore essential.
- We have to deal with a large variety of ship types, so we cannot set up a parametric description specific for one class of ships.
- The time available for the wave resistance minimisation is usually 1-2 weeks or less. Thus we need a practical, effective and efficient method to optimise the hull form. Reproducing
the initial design by a fully parametric description or formulating all geometric constraints explicitly would be an undesired preceding step.

This has led us to the present approach, which is characterised by its use of completely general parametric deformations of the initial hull form, in which many constraints are already inherently taken into account; by a focus on parameters that have a direct connection with flow properties we want to address; by efficient evaluation of the objectives using free-surface potential-flow (or free-surface RANS) solvers; and by the use of surrogate-based global optimisation for the frequent multi-objective optimisation tasks. These main components will now be described and some examples discussed.

HULL FORM PARAMETRISATION

Perhaps the least settled aspect of a hull form optimisation is the parametric description of the hull form and its changes. Many possibilities have been proposed. In [8] it is pointed out that the success of an optimisation hinges upon the ability to reproduce by the parametric description, the various shapes that an experienced naval architect would design. This has not always been the case for methods proposed.

The first choice to be made is, whether one tries to represent the entire hull form by a parametric description, or applies parametric deformations to an existing hull form. The former option may seem attractive; but it takes a very large number of parameters to be able to describe whatever ship form, and it remains approximate in most cases. Once the initial hull form provided by a yard would have been reproduced by the parametric description, a part of the parameters could be varied to make modifications; but selecting these and defining all related constraints seems a time-consuming and insatisfactory process. Therefore, we choose the option of parametric deformations of an initial hull form.

Several possibilities have been proposed for this. Some use generic deformations, such as Lackenby shifts [6], modifications of section shapes by Fourier components [9], or additive polynomial surface patches [5]. The advantage of these is that they can quickly be applied and could give a first impression of possibilities for improvement; but they are unlikely to lead to a final, detailed hull form satisfying all requirements. More general are Free-Form Deformation [4,8] or movement of some freely chosen control points followed by Radial Basis Function interpolation of the hull surface [6]. Still these have certain limitations [8]. Also, to apply most of these methods to generate a particular hull form feature would require practical experience. That skill would need to be developed just for this purpose, which we believe is a disadvantage.

However, using a CAD package to create a modification to an existing hull form is a skill that is available. An experienced CAD engineer makes a modification in very limited time, e.g. less than an hour. The resulting hull form is smooth and faired, feasible, has the desired displacement and satisfies the required hard points and some other constraints. At the same time, we have complete flexibility in the type of modification.

Therefore, we use the CAD system to create N deformed versions of the initial hull form. Each represents one particular deformation mode, selected so as to be more or less independent from the others, and normally defines the largest deformation we want to consider. The N deformed versions, together with the original hull form, define an N-
dimensional design space of hull forms, of which the NURBS control point positions are interpolated between those of the original and the deformed hulls. The interpolation factors are the N parameters of the hull form family. The hull form variation thus is a parametric blending or morphing of hull forms. There is complete generality, as long as all hull forms are generated by a similar control point network. Fig.1 gives an example of shapes obtained.

Figure 1 Some bulbous bow shapes generated with 2 parameters: one for contour shape, one for bulb width. The examples have parameter values (0,0) (initial design), (0.5,0.5), (1.0,1.0) and (1.0,1.5)

The most deformed hull forms can be created in the CAD system’s fairing mode, or also by additional tools such as an FFD-like method [3] or Lackenby shifts. The amount of work is limited, and is easily compensated by the good quality of the hull forms, the limited number of parameters needed for meaningful hull form changes, and the reduction of the number of constraints needed. E.g. if a local bulb modification to an existing hull form is to be designed, we create modified designs which all match the existing hull form at the position required, and all intermediate shapes will also do so. Similarly, other constraints such as keeping equal displacement and LCB, or trivial issues such as keeping a flat bottom and sides, are incorporated in the deformed shapes and need not be explicitly prescribed as constraints, as they might for generic shape variations.
Once the design space has been created, a hull form and hull panelling can be generated in seconds by a batch command to the CAD system, with the desired parameter values as input.

THE DESIGN SPACE

The freedom to define hull form variations allows, and requires, to make a judicious choice of which to include in an optimisation. We want the parametric changes to be geometrically independent, or orthogonal: a particular hull form should be described just by a single combination of parameters. But also a level of independence of their physical effect is desirable: it would be best if a given trend of the flow field or wave pattern is represented by one or few parameters.

To illustrate, let us consider the action of a bulbous bow. This is strongly determined by the underpressure it creates at its sides, so by the streamline curvature, and its longitudinal location [1]. Suppose a bulbous bow would be described parametrically in terms of the sectional area curve, section shape parameters and a bulb length parameter; these would have just an indirect relation with that curvature, and the response of the bow wave making to these parameters might be an irregular function with more local extrema. Instead, we would typically generate 2 extreme shapes, one with a different bulb length but similar curvature, the other with the same length but different width and curvature. These two parameters would map to the two main physical trends of the bulb action.

In general, we aim at basing the hull form variation parameters on physical effects, and describing those effects with as few and as meaningful parameters as possible. This differs from a purely geometrically-oriented hull form description or deformation, including the Principal-Component-Analysis based method proposed in [10] which seeks to represent with few parameters most of the geometric variability. Instead, our approach is unsystematic but aims for maximum objective variability. Often we assess the sensitivity of the objectives to the parameters beforehand by making computations for some separate hull forms, or systematic hull form variations with subsets of parameters using the RapidExplorer system [3].

Consequently, due to the choice of deformation modes based on hydrodynamic insight, and due to the stage of the design that we typically work in, we often have a limited number of free parameters in the optimisation.

EVALUATION OF THE OBJECTIVE FUNCTIONS

The hull form parametrisation described is being applied with different flow solvers. For viscous-flow related optimisation, we mostly use the Parnassos code, a fast free-surface RANS code [11]. This is also the method of choice if reduction of the stern wave making, for a less slender vessel, is desired. For minimising the required power of ships it is coupled with a propeller representation. Also for this method the optimisation approaches described here have been used and are being further deployed [12].

In this paper however, we focus on the use of free-surface potential flow codes, RAPID [13,14] in particular. This panel method computes the wave pattern and wave resistance by solving the steady nonlinear free-surface potential-flow problem iteratively. After convergence the complete inviscid kinematic and dynamic boundary conditions are satisfied and the dynamic sinkage and trim are incorporated. Rankine source panels are located on the
hull surface, and at a small distance above the wave surface. Panel distributions are automatically adjusted between iterations, as are dynamic trim and sinkage. In each iteration, the boundary conditions are imposed in collocation points on the hull and on the last free-surface iterate, and the resulting system of equations is solved by a preconditioned GMRES solver. Usually about 3000 panels on the hull and 5000-20000 on the free surface (per symmetric half) are used, dependent on Froude number. Convergence is typically in 8 to 20 iterations. The RAPID code is usually run on a standard desktop PC and takes from 1 to 10 minutes for the entire computation for one speed.

This code is in continuous practical use in ship design, at MARIN and elsewhere, since 1994. It yields accurate results for a large class of ships, for the wave making from the bow and forebody, fore and aft shoulders, and, for slender vessels, also for the transom stern. For fuller hull forms however, viscous effects play a significant role in the stern wave making and larger deviations occur.

Wave resistance is evaluated both by integration of pressure forces over the hull, and from wave pattern analysis based on a set of transverse wave cuts aft of the ship [14]. The former method is slightly less suitable for optimisation due to some numerical noise from variations in the hull panelling, so mostly the latter value is used as the objective; optionally augmented by a viscous-resistance estimate based on an estimated form factor and a plate friction line. This disregards any variations of the form factor with the hull form variation, which of course is not precise for larger afterbody variations.

THE OPTIMISATION METHOD
After some experimentation with other codes, we have adopted the Dakota package [15], an extensive collection of optimisation tools developed by Sandia National Laboratories. Of its many options, we describe here an approach we have found suitable for our needs so far.

For single-objective optimisation problems, e.g. minimising wave resistance for a single speed and draft, gradient-based methods can work, but tend to be sensitive to numerical noise and some experimentation is required. Different starting points may need to be used to find a global optimum. But in many cases we have to address multiple conditions, e.g. different speeds and drafts, and we resort to other methods. The formal way to balance different conditions in multi-objective optimisation is a major asset for such problems.

For a global multi-objective optimisation, genetic algorithms are a robust choice. However, they often require thousands of objective evaluations, as successive generations converge just slowly to the optimum. Applying a genetic algorithm directly to the solver we found too inefficient. But for the choice of parameters as we make, the dependence of the objectives on the parameters is usually fairly smooth. In that case, surrogate-based methods using response surfaces can work very well. Such methods have also been adopted in [6,16] but without the successive improvement that we apply.

We start with generating a Design of Experiments (DoE), a set of hull form variations spread in a particular fashion over the design space; e.g. a Latin Hypercube Sampling. The number of variations can be quite limited, but a too small number does not help in a later stage. E.g. for a 5-parameter family, we have used 100 variations. For each, the potential-flow code is run for all conditions. Response surfaces are then generated, algebraic functions that
interpolate or approximate the objectives as a function of the design parameters. Next, the genetic algorithm is run to carry out the multi-objective optimisation, but it uses evaluations of the response surfaces only, no direct potential-flow computations. Consequently this takes negligible calculation time. For this surrogate-based optimisation, the choice of the actual optimiser is therefore immaterial for efficiency. Any robust global optimiser would do, and our rather conventional choice of a genetic algorithm is no drawback.

The output of this stage is a Pareto-optimal set of variants, but based on the response surfaces. Whether this is a sufficiently accurate approximation needs to be checked. In practice we have noticed RMS errors of 1-5% in the estimated objective functions for these points, so the Pareto front may need to be determined more precisely. Therefore, next the potential-flow code is run for a selection of points on the estimated front, typically 10-20 hull forms in our case. The results of these computations, generally deviating from the objectives interpolated on the response surfaces, are then added to the DoE and the response surfaces are updated. Thus they become more accurate where it matters, i.e. in the vicinity of the Pareto front. We reapply the optimiser using these updated response surfaces, get an improved Pareto front, and may continue this iterative process if needed. In this way, the effect of the chosen size of the DoE and the accuracy of the initial response surface should play no role in the final result, and the Pareto front can be derived extremely efficiently.

However, one needs to survey the process as some of the settings make a difference. A choice to be made is the type of response surface. Dakota supports a variety of options, among which quadratic and cubic polynomial surfaces, or Kriging. An example of how these compare is shown in Fig. 2. Clearly there are significant differences; although in this case all response surfaces indicate the same optimum values of parameters. It is, therefore, essential to carry out the step to reevaluate points along the Pareto front to get a true value, and to update the response surfaces and the estimated front. Fig. 2 illustrates the resulting update of the response surface, which is not dramatic but significant.

We also note in Fig. 3 that for one of the intermediate values of the third parameter, the initial response surface has a large deviation at the right front corner, suggesting very low resistance values for those parameters. This appears to be a spurious result, which is only as extreme for the cubic polynomial surfaces. In this case the optimiser does not go to that corner (as a result of the three other objectives taken into account), therefore the updated surface still has the same feature; but in other cases it might cause an erroneous estimated first Pareto front which would disappear in next iterations. On the other hand, a possible pitfall would be if the initial response surfaces overlook a genuine optimum, subsequent refinements occur at another place and this true optimum is never detected. It illustrates that there is a tradeoff in the choice of the size of the DoE.

So far the Kriging response surfaces seem better behaved than the cubic polynomial surfaces. However, if the iterative improvement of the surrogate is continued for more steps, there will be many closely-spaced points near the front which may cause deviations in the Kriging surface, possibly leading to nonconvergence of the front. Kriging is also sensitive to addition of new points due to the Maximum Likelihood Estimation procedure, see [17].
Figure 2 Example of initial response surfaces. For a fixed value of one parameter, the response against 2 other parameters is shown, as derived from the initial Design of Experiments of 40 variants in a 3-parameter space. Top to bottom in the far left corner: cubic polynomial, Kriging, quadratic polynomial.

Figure 3 Cubic polynomial response surfaces, before (with mesh) and after update; the markers indicate the added points along the Pareto front.
This surrogate-based optimisation works quite well for our purposes. However, one should keep in mind that we have made particular choices for the parametrisation, with parameters chosen to be related with physical effects, not with geometrical features only; thereby with limited numbers of parameters involved in the optimisation and hopefully, a relatively clear relation with the objectives. This probably contributes to the adequacy of the response surfaces and of the surrogate-based optimisation based on those.

**APPLICATIONS**

While these methods have been used in several practical design projects, we consider one hypothetical example here, for which more details can be shown. The case at hand is a product carrier as used in the EU-project ‘Streamline’ [11]. Fig. 5 (bottom) shows the computed wave pattern and hull pressure distribution for a speed of 14 kn (Fn = 0.237). We notice rather substantial wave making, with a high bow wave and diverging waves radiated out; a pronounced fore shoulder wave trough, followed by transverse waves along the hull; a substantial transverse wave system aft of the hull, and slight aft shoulder waves. The graph for 11 knots (Fig. 6, bottom) shows much less wave making, but a fore shoulder wave trough at the same position, again transverse waves along the hull, and a diverging bow wave system. Therefore, what needs to be improved for this ship is at least the position and curvature of the fore shoulder; the bulbous bow action, which is now insufficient to reduce the high bow wave; and possibly, some improvements at the aft shoulder, to reduce the steep wave slope towards the transom for 14 kn. While probably too much affected by viscosity for a potential-flow code, we also try to change the transom stern to reduce the transverse wave system aft.

We define 5 parameters:

- A softening and aft shift of the fore shoulder;
- A parameter changing the bulbous bow contour to a more horizontal shape, with simultaneous increase of its length;
- A parameter increasing the width and waterline curvature of the bulbous bow;
- A parameter that shifts the aft shoulder somewhat forward;
- A parameter that lifts the transom slightly and makes the waterline endings more horizontal.

The two parameters for the bulbous bow produce a family of shapes, some of which have already been illustrated in Fig.1. Otherwise these 5 parameters are each related with an aspect of the wave making, and we expect just limited interaction between them.

In this 5-dimensional design space, we generate a DoE of 100 hull form variations, based on Latin Hypercube Sampling. For these rather low Froude numbers the potential-flow computations require a panelling of about 11000, but still a computation time of just 7 min per hull form for 2 speeds, on a single PC; so the DoE can be completed overnight.

In Fig.4 we show the results in a Pareto plot. They appear to be spread around the initial hull form. Response surfaces are then generated, using Kriging, and the genetic algorithm is run to generate a Pareto front, indicated by the open red markers in the figure. The corresponding hull forms are then evaluated by the potential-flow solver, producing the results indicated by the full red markers. Clearly, there is a small deviation between both:
the RMS error of the response surface estimates for both objectives in these points amounted to 3.2 and 3.4%, respectively. This is 1/5 of the total improvement relative to the initial hull, so refinement is desired. Subsequently, new response surfaces are built taking into account the DoE plus the new points; optimisation is done on these new surfaces, etc. The blue markers show the Pareto fronts after some of these iterations, for which the estimate and the true values are near identical, errors having been reduced by a factor of 10. In fact, the first true front was already the final one; which means that we had found the optimal hull forms with just 2 * 127 potential-flow calculations.

As appears, about 19% wave resistance reduction has been achieved for 11 knots, 14% for 14 knots with the hull form adjustments allowed here. This amounts to some 6-7% of the total resistance. Figs 5,6 show the wave pattern for the original design and one of the hull forms on the Pareto front, for 14 and 11 kn. There is a clear reduction of the transverse waves along the hull, of the bow wave crest and diverging wave system; and a small reduction of the aft shoulder wave. The transom modification has not really worked though. Still, we conclude that the design space defined was effective, and a significant improvement has already been obtained in very little time (1-2 days).

Various other practical applications have been done. One design question concerned a ferry with a demand for minimum wave resistance at a given higher speed, but a radiated bow
Figure 5 Wave pattern at 14 kn, for optimised (top) and original hull form. Vertical scale 2 times magnified.

Figure 6 Wave pattern at 11kn, for optimised (top) and original hull form. Vertical scale 2 times magnified.
wave amplitude that had to be reduced to a given upper limit at a lower speed. In that particular case, the problem was successfully solved as a two-objective problem, minimising the high-speed wave resistance and low-speed wave amplitude. This produced a Pareto front, and the design selected was the point at that front with a low-speed wave amplitude just satisfying the imposed limit (with some margin).

In another project, a containership had to be designed for an operational profile, consisting of 4 drafts and 3 speeds, for which the relative time spent in each condition was provided. We have condensed this to 4 conditions that covered most of the total time, and defined a design space based on extensive preceding sensitivity studies. The optimisation aimed at reducing the sum of wave resistance and estimated viscous resistance, using a constant form factor for simplicity. It led to significantly improved wave making for all 4 conditions. Importantly, the multi-objective optimisation and identification of a Pareto front, followed by selection of the desired hull form using a weighting of the 4 conditions based on time spent and estimated fuel consumption, formed a systematic answer to the request to optimise for an operational profile.

CONCLUSIONS

A practical approach has been presented for minimising ship wave resistance, using a free-surface potential-flow code connected to a CAD system and the Dakota optimisation package. The system is primarily directed at a relatively late stage in the hull form design, when main dimensions, main coefficients and LCB are already known but otherwise the hull form is to be finalised. Hull form variations are obtained by a blending of the original hull form and a number of modified hulls that determine the main axes of the design space. These are simply generated using the CAD system and thus provide complete flexibility. Consequently, there is no need for preset generic hull form modifications as are often proposed; and several geometric constraints can already be taken into account.

The flexibility of defining hull form modifications is exploited by choosing parameters that are directly related with a physical aspect that is to be improved. Thereby, the designer’s skill and hydrodynamic knowledge can be exploited. Also it typically results in a relatively small number of design parameters, and a fairly regular and smooth relation between design parameters and objectives. As a result, adequate response surfaces can be generated based on a rather limited sampling, and surrogate-based global optimisation then appears to works very well. Iteratively updating the response surfaces was found essential.

An example of a 2-objective problem in a 5-parameter space could be solved with just 127 hull forms directly evaluated. The experiences obtained are also being applied in combination with free-surface viscous-flow computations [12], for which an efficiency gain is even more important.

These methods are now used increasingly in practical ship hull form design projects at MARIN, and offer a step forward in the efficiency and effectiveness of the design process.

ACKNOWLEDGEMENT

This research was funded from the TKI-Allowance of the Dutch Ministry of Economic Affairs. The support is gratefully acknowledged.
REFERENCES
IMPROVED MODELLING OF SHEET CAVITATION DYNAMICS ON DELFT TWIST11 HYDROFOIL

Guilherme Vaz\textsuperscript{1}, Thomas Lloyd\textsuperscript{1} and Arun Gnanasundaram\textsuperscript{2}

\textsuperscript{1}MARIN
Haagsteeg 2, Wageningen 6708 PM, The Netherlands
e-mail: \{g.vaz; t.lloyd\}@marin.nl

\textsuperscript{2}TU Delft
Kluyverweg 1, Delft 2629 HS, The Netherlands
e-mail: A.K.Gnanasundaram@student.tudelft.nl

Key words: Delft Foil, Cavitation, RANS, Reboud-correction, DDES, ReFRESCO

Abstract. In this paper, unsteady viscous-flow cavitation predictions for a 3D hydrofoil are performed using three different approaches: 1) a pure RANS method; 2) a RANS method including an eddy-viscosity “Reboud” correction; 3) a DDES Scale-Resolving-Simulation approach. Both wetted and cavitating flow conditions are analysed and compared with experimental data. The accuracy of these approaches is scrutinised in terms of integral quantities, cavity dynamics, different background principles, and the influence of grid refinement. Low numerical uncertainties have been obtained for wetted flow conditions, but a somewhat large deviation on the wing loading has been observed when compared with experimental results. For the cavitating flow case, the RANS calculations do not accurately simulate the cavity dynamics. The RANS-Reboud correction improves the fidelity of the calculations at increased numerical demands and decreased robustness. The DDES approach leads to improved dynamics, slightly less accurate cavity shedding mechanism, at lower computational cost. The cavity extents are underpredicted for all methods and conditions used, when compared with the available experimental data.

1 INTRODUCTION

Cavitation occurs when the static pressure of the liquid drops below its corresponding saturation pressure leading to the formation of vapour. This often occurs near the leading edge of the blades of devices such as propellers and pumps. Part of this vapour then breaks off from the leading edge, convects downstream, and undergoes condensation. The process of vapour shedding imposes fluctuating loads on the blades, while the collapse of the vapour leads to erosion of the surface of blades. Associated pressure fluctuations and noise generation are also often an issue. Hence the susceptibility of a propeller design to cavitation must be understood during the design stage, which has led to research attention focussing on the computational modelling of cavitation. It has been noted however that the interactions between the vapour and the tur-
bulence structures reveal some complexities which make it challenging to capture the cavitation mechanism accurately using common engineering numerical modelling approaches [1].

Cavitating flow simulations are very sensitive to the turbulence and cavitation models used. Traditional Reynolds-averaged Navier-Stokes (RANS) models tend to overpredict the turbulent viscosity in the cavity closure region, dampening the turbulent interactions that promote shedding. This has been well established in [2, 3]. As put forth in [4], a way to overcome this deficiency is by using *ad hoc* corrections to reduce the turbulent dissipation in those regions, allowing the vapour to grow and shed from the leading edge. Though the effectiveness of this correction in promoting shedding with different turbulence models have been demonstrated [1, 3, 5], the influence of grid resolution effects and discretisation errors on the cavitation solution have not received as much attention.

The failure of standard RANS models to accurately capture the unsteady phenomenon of cavitation is not surprising, since these were not originally designed for multiphase unsteady flows. On the other hand, Scale-Resolving Simulations (SRS) are expected to give a better prediction of reality. Large-Eddy Simulations (LES) and Delayed-Detached Eddy Simulations (DDES) restrict the extent of modelled turbulence, thereby reducing the modelling error. SRS performed in [3], [6] and [7] report the improvement in the predictions and show increased cavity length and shedding frequencies that match more closely with experiments than RANS. [3] and [8] also provide comparisons between RANS and SRS, but limit themselves to a quantitative comparison of the mean lift coefficient, vapour volume, and shedding frequency. In this paper, we aim to analyse the differences between the application of SRS (more specifically the DDES formulation), RANS and RANS including eddy-viscosity correction for unsteady cavitation predictions. We analyse their numerical requirements, different background principles with respect to cavitation dynamics, and the influence of numerical errors on their working behaviour.

The chosen test case is the well-known Delft ‘Twist11’ Foil, for which experimental results from [9] and [10] are available for use as validation material, while comparison can also be made to numerous computational results from the open literature. The numerical approach taken, the discretisation schemes, and a brief description of the solver used in this work is covered in Section 2. Details of the grids generated for the study, the computational domain and numerical settings adopted are given in Section 3. Results of the wetted and cavitating flow simulations are included in Section 4. Finally, concluding remarks are made in Section 5.

## 2 SOLUTION METHOD

### 2.1 Flow solver

The governing equations were solved using a finite volume viscous-flow CFD code ReFRESCO (www.refresco.org) developed at MARIN, Technical University of Delft and several other universities around the world. The equations are solved in a segregated manner and the coupling happens iteratively using a SIMPLE-type algorithm. A second order QUICK scheme was used for discretisation of convective fluxes in the momentum equation, with a flux limiter activated to provide numerical stability. For fluxes in the turbulence equation and the transport equation for cavitation, a first-order upwind scheme was used. A second-order implicit time marching scheme was used.
2.2 Turbulence and cavitation modelling

For RANS simulations the $k - \sqrt{kL}$ model for turbulence [11] was used. This model predicts lower levels of eddy viscosity ($\nu_t$) than the more commonly-used $k-\omega$ SST model, and has been shown to improve prediction of cavitation dynamics for propellers [12]. For some simulations, this model was combined with the viscosity correction proposed in [4] (and thereafter referred as Reboud correction), which reads as

$$f(\alpha) = \rho_v + \left(\frac{\rho - \rho_v}{\rho_l - \rho_v}\right)^n (\rho_l - \rho_v) = \rho_v + (1 - \alpha_v)^n (\rho_l - \rho_v), \quad n \geq 1,$$

where $\alpha$ is the volume fraction, $\rho$ is the mixture density and the subscripts $l$ and $v$ refer to liquid and vapour values respectively. The parameter $n$ must be chosen, with the authors of the correction recommending that 10 be used. The function $f(\alpha)$ is used to correct the magnitude of the eddy viscosity predicted by the turbulence model $\mu_t = \mu^* f(\alpha)/\rho$, with $\mu_{eff} = \mu + \mu_t$ being the effective mixture viscosity used in the diffusion term of the momentum equations.

Scale resolving simulations (SRS) were performed using Delayed Detached Eddy Simulations (DDES) based on the $k - \omega$ SST model [13]. DDES aims to resolve turbulent structures in separated flow regions and switches to RANS in regions where the mesh size is not large enough to capture the small scale structures, as well as in near-wall regions (enforced). The switch between RANS and LES for DDES is determined based on a localised length scale ($l_t$) given as

$$l_t = l_{RANS} - f_d \max(l_{RANS} - l_{LES}, 0),$$

where $l_{RANS}$ is the turbulent length scale of the $k - \omega$ SST model, $f_d$ is a blending function and $l_{LES} = C_{DES} \Delta$ is the LES length scale based on the local cell size $\Delta = \max(\Delta x, \Delta y, \Delta z)$. $l_t$ is then used to decrease the destruction of $k$ of the SST model, and therefore decrease $\mu_t$.

Cavitation was modelled by considering a homogeneous mixture of liquid and vapour bubbles. A transport equation for the vapour volume fraction $\alpha_v$ is solved, with the source term based on the square root of the difference between the local and saturation pressure according to [14] (Sauer model). The slight modifications to the source term proposed by [15] were also used.

3 COMPUTATIONAL SETUP

3.1 Computational domain and boundary conditions

The Delft Foil is a twisted hydrofoil with a NACA0009 profile and has a spanwise varying angle of attack (AoA) from 0° at the sides to 11° at the midspan. The foil has a chord length ($c$) of 0.15 m, and is placed at an AoA of $-2^\circ$. The domain inlet was located $2c$ upstream of the leading edge (LE) of the foil, and extends to $5c$ downstream. The $x$-axis was aligned with the flow, while the $y$-axis extended in the spanwise direction from 0 to $b$ (the foil halfspan, equal to $c$) with a symmetry condition imposed on the midspan plane. The top and the bottom walls were both located one chord from the foil. A no-slip condition was imposed on the foil surface, while the side walls were treated as slip walls. An inflow boundary condition was specified at the inlet, while at the outlet a fixed zero pressure was used. The computational domain is shown in Figure 1.
Table 1: Summary of grid statistics for all halfspan grids.

<table>
<thead>
<tr>
<th>grid</th>
<th>no. of cells / M</th>
<th>target $y^+_{max}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>G2</td>
<td>4.50</td>
<td>0.40</td>
</tr>
<tr>
<td>G3</td>
<td>2.10</td>
<td>0.50</td>
</tr>
<tr>
<td>G4</td>
<td>1.30</td>
<td>0.65</td>
</tr>
<tr>
<td>G5</td>
<td>0.42</td>
<td>0.80</td>
</tr>
</tbody>
</table>

3.2 Grids

Four block structured grids were generated using GridPro, by successively coarsening the finest grid by a factor of 1.25. The total number of cells and targeted maximum normalised wall-normal cell height $y^+_{max}$ are listed in Table 1. The resulting grids are geometrically similar allowing systematic analysis of the effect of grid refinement on the flow solution.

In order to adapt the grids somewhat for cavitation prediction, cells were concentrated on the suction side of the foil, including in the region where shed cavitation was expected, as well as towards the midspan of the foil (symmetry plane of the domain). It was also expected that this topology would help in the resolution of shed vortices when performing DDES. Figure 2 clearly shows the grid density in this region.
3.3 Computational settings

The chosen test conditions correspond to those frequently seen in the literature for numerical computations of the geometry. Velocity at the inlet was prescribed as $U_0=(6.97, 0, 0) \text{ m/s}$, with inlet values for the turbulence quantities set in order to give a turbulence intensity of 1%. The pressure at the outlet was fixed at $p_0 = 0 \text{ Pa}$. Wetted flow steady simulations were performed by setting $p_v$ to avoid the formation of vapour. Both RANS ($k - \sqrt{k}L$ based) and DDES ($k - \omega \text{ SST}$ based) were performed. In steady attached flow, these DDES calculations are almost pure RANS. The solution was considered to be well converged once the $L_\infty$-residuals reached $1 \times 10^{-9}$. A wetted flow solution served as starting point for cavitating flow simulations with cavitation introduced in steps by adjusting the vapour pressure $p_v$ to achieve the correct cavitation number, which was $\sigma = 1.07$, with $\sigma = 2(p_0 - p_v)/(\rho_l U_0^2)$.

Both liquid and vapour values for the fluid densities and dynamic viscosities were specified viz. $(\rho_l, \rho_v) = (998, 0.024) \text{ kg/m}^3$ and $(\mu_l, \mu_v) = (100.2, 1.02) \times 10^{-5} \text{ kg/m/s}$. For the cavitation model, the number of nuclei per unit volume was set to $10^8$ while the minimum bubble radius was restricted to $1 \times 10^{-5} \text{ m}$. Finally, the Reboud parameter which controls the magnitude of the eddy viscosity correction was chosen as $n = 4$, much lower than typically used in the literature. This value was used due to the fact that the eddy viscosity level for the $k - \sqrt{k}L$ model was expected to be several times lower than would be predicted using the $k - \omega \text{ SST}$ model.

RANS cavitating-flow calculations were performed using a normalised timestep $\Delta t^* = \Delta t U_0/c$ of $2.3 \times 10^{-3}$, while $\Delta t^* = 7 \times 10^{-4}$ was used for DDES simulations. The lower timestep for DDES was chosen aiming to improve the temporal resolution of the scale-resolved part of the flow such that the affect of grid size was more important than timestep. For RANS computations with Reboud correction, the number of outer loops had to be increased with the number of grid cells in order to achieve sufficient iterative convergence within each timestep.

4 RESULTS & DISCUSSION

4.1 Wetted flow simulations

Comparison of wetted flow simulations are made in terms of the lift, drag and pressure coefficients, defined as $C_L = 2L/(\rho_l cb U_0^2)$, $C_D = 2D/(\rho_l cb U_0^2)$ and $C_p = 2(p - p_0)/(\rho_l U_0^2)$. Here $L$ and $D$ are the forces perpendicular to and parallel to the inflow direction, and $cb$ is the foil planform area. An uncertainty analysis using the method proposed in [16] was performed to assess the discretisation uncertainty ($U$) for the RANS calculations. Note that for DDES, even in steady attached flow, the LES/DDES branch is activated in the wake of the wing. Moreover, in DDES the modelling error is entangled with the numerical error due to the grid-dependent LES filter, and therefore the V&V method of [16] cannot be applied.

The differences in $C_L$ between the finest and the coarsest grid in RANS and DDES simulations are less than 0.6% and 0.8% respectively. The corresponding values for $C_D$ are even smaller for both turbulence models. For the finest grid (G2) the RANS results are 0.425 for $C_L$ and 0.0146 for $C_D$, while for DDES these are 0.428 and 0.0143 respectively. The associated uncertainty $U$ for the RANS calculations is 1.0% for $C_L$ and 1.5% for $C_D$. Even if it has been recognised in
The non-dimensionalised pressure on the foil along the midspan plane is plotted alongside experimental data from [9] in Figure 3. A good agreement is observed between the two data sets, while the sensitivity of the surface pressure distribution to grid density is seen to be small.

4.2 Cavitating flow simulations

4.2.1 Flow fields

Figure 4 illustrates a general overview of the cavity layout (represented by an iso-surface value of $\alpha_v = 0.1$), together with limiting streamlines, and contours of eddy-viscosity, for the three approaches here employed. Figure 5 shows axial velocity and off-plane vorticity fields at the symmetry plane (halfspan), together with a vapour contour of $\alpha_v = 0.1$.

These instantaneous flow fields show that the cavity predicted by a pure RANS solution is small. Also, the eddy-viscosity exhibits high values inside and downstream of the cavity as well as in the wake at the symmetry plane, the most highly-loaded section of the wing. The limiting streamlines show backward flow inside but also downstream of the cavity. The flow is steady, mostly two-dimensional and there is a small re-entrant jet, but no side re-entrant jets. With RANS-Reboud the cavity sheds and the flow is highly unsteady. Inside the cavity and shed vapour pockets the eddy-viscosity is reduced; but around these structures, or associated shed vortical structures without vapour, the eddy-viscosity values are high. The limiting streamlines inside the cavity are more chaotic, and both side and longitudinal re-entrant jets can be seen. Figure 5 shows more clearly the re-entrant jet present in both approaches, and the more complex structures both in terms of vapour pockets and vortices in the DDES results.

---

Corroborated by Evert-Jan Foeth [17].
Figure 4: Cavity extent, limiting-streamlines and eddy-viscosity field. Instantaneous values. RANS (top-left), RANS-Reboud (top-right) and DDES (bottom).

Figure 5: Axial velocity field (left), off-plane vorticity field (right) and cavity contour ($\alpha_v = 0.1$). Instantaneous values on symmetry plane. RANS-Reboud (top) and DDES (bottom).
4.2.2 Effect of grid resolution

Figure 6 shows values of $C_L$, Strouhal number $St$ (defined as $St = f c / U_0$ where $f$ is the main shedding frequency) and total vapour $V_v$ obtained for the current simulations and grids, together with results available in the literature. Figure 7 presents the time history of lift and vapour vs the grids used. Figure 8 shows the effect of the grid resolution on the cavity layout, limiting streamlines and eddy-viscosity values.

As also seen in the previous section, the RANS simulations give a grossly underpredicted cavity, which only extends until about 20% of the chord ($l/c = 0.2$), and remains attached to the foil surface. The dynamics does not improve with grid resolution; in fact, the results reach (almost) steady state with $V_v$ becoming constant and reducing with increasing grid resolution. Application of the Reboud correction has a large effect on the flow solution. A value of $n = 4$ almost doubles the length of the cavity, and the lift also increases. The time histories of $V_v$ and $C_L$ in Figure 7 show that significant cavitation dynamics also result, yielding a $St$ of around 0.7, as in the experiments. Grid refinement has a somewhat non-monotonic behaviour both in terms of on lift and total vapour volume, showing the existence of possible numerical issues. As stated previously, the number of non-linear outer-loops had to be increased significantly for RANS-Reboud with increasing grid resolution. Nevertheless, in general, one can say that the cavities are larger and have a larger volume with increasing grid resolution. The peaks seen in the lift curve are due to the condensation/collapse phase of the cavity, and have been reported several times in the literature by other authors, specially when using the Sauer cavitation model.

![Figure 6: Mean $C_L$, $St$ and $V_v$ variation with total number of grid cells for RANS, RANS-Reboud, (D)DES and LES cavitating flow simulations. Crosses denote results from the present study, while circles show results from literature.](image)

The DDES results yield slightly higher $C_L$ but lower $St$ and vapour volumes than those using RANS-Reboud. The time histories of the lift and vapour volume show a more monotonic and well-behaved convergence with grid refinement. In DDES, the finer the grid, the more turbulent structures are being solved, and the larger the dynamic behaviour of the solution. Also, it is interesting to note that for the coarsest grid G5 DDES leads to steady results. On the contrary, for the same grid, even a “simpler” RANS solution (not shown here) leads to shedding
Figure 7: Time histories of \( C_L \) and \( V_v \) for RANS-Reboud and DDES cavitating flow simulations, showing effect of grid refinement.

and dynamic behaviour. Figure 8 shows that grid refinement reduces the eddy-viscosity and increases the vapour pockets shed, in both RANS-Reboud and DDES cases.

The comparison of the present results with the ones available in the literature show similarities on \( St \), with values around 0.70. However, for \( C_L \) the spread is larger with variations more than 10%. It is also interesting to note that the total vapour volume is rarely reported in the literature, and that the grids used in the current work for the halfspan wing are similar in terms of resolution to the ones used by some authors for the complete wing.

4.2.3 Cavity dynamics

In order to further study the working principles of the Reboud correction and DDES on cavitating flow dynamics, Figure 9 presents the instantaneous eddy-viscosity field, Reboud correction \( f(\alpha)/\rho \), with \( f(\alpha) \) defined by Equation 1, and DDES activation region \( 1 - \frac{l_t}{l_{RANS}} \), with \( l_t \) defined by Equation 2, at the symmetry plane computed using the finest grid G2.

The eddy-viscosity levels are significantly reduced in the DDES calculations for the complete plane, while the Reboud correction decreases these levels only inside the cavitation/vapour areas. The lower left part of Figure 9 shows the values of \( f(\alpha)/\rho \) for the same time instant, showing that this correction is only active within vapour zones. The lower right part of Figure 9 shows that even for G2, with 4.5M cells for the halfspan wing, the values of \( 1 - \frac{l_t}{l_{RANS}} \) are still too high (always larger than 0.5), and that due to the DDES intrinsic boundary-layer shielding, the LES mode is not activated within vapour areas inside the boundary-layer. This affects the re-entrant jet dynamics and the associated overall shedding mechanism. It is believed that grid refinement, especially in the longitudinal and transverse directions will slightly improve this issue, but not solve it. Other SRS models, for instance XLES, IDDES or PANS, which are allowed to solve turbulence anywhere in the flow field are needed, and the authors are of the opinion that for these same reasons wall-modelled LES models will not be completely effective for this type of cavitating flow.
Figure 8: Cavity extent, limiting-streamlines and eddy-viscosity field. Instantaneous values. RANS-Reboud (left) and DDES (right); G4 (top) and G2 (bottom).

Figure 9: Eddy-viscosity $\mu_t$ (top), Reboud correction $f(\alpha)$ (bottom-left) and DDES activation $1 - \frac{b_{\text{RANS}}}{b}$ (bottom-right) fields. Instantaneous values on symmetry plane. RANS-Reboud (left) and DDES (right).
4.2.4 Comparison with experiments

Figure 10 shows the time-averaged $C_p$ on the foil surface at the symmetry plane compared against experimental values [17]. One can clearly see that the RANS approach underpredicts the cavity length (and its dynamic behaviour) and that both RANS-Reboud and DDES deliver considerable improvements. There is a significant under-pressure area ($\sigma = 1.07$ in this case) both in the numerical calculations and experiments. Calculations underpredict the cavity length using all numerical approaches. Note that the predicted lift, even in wetted flow, is also too low when compared with the experiments, and therefore some uncertainty on these results is expected to be caused by the lower loading of the wing in the numerical calculations.

Finally, Figure 11 illustrates for two time instants the comparison of the predicted cavity layouts with the experimental data available in [17]. These pictures demonstrate that the RANS-Reboud correction predicts the well established cavity re-entrant jet formation and upstream impinging flow, which is the origin of the cavity dynamic behaviour for this test-case. On the contrary, the DDES approach does not show this, and a more bulk vapour detachment external to the boundary-layer is seen. When compared with the experimental data, both numerical simulations underpredict the cavity length, volume and cloud cavitation formation. One can also notice that for these two instants the experiments show some asymmetry. While for RANS-Reboud the use of the complete wing is not expected to lead to different results, for an SRS approach like DDES this might have a measurable influence on the results.

5 CONCLUDING REMARKS

In this paper, we have analysed the differences between the application of a RANS, RANS including an eddy-viscosity (Reboud) correction and a DDES SRS approach for unsteady cavitation predictions on the 3D Delft “Twist11” Foil. We have examined these methods’ quantitative accuracy both in terms of integral quantities and cavity dynamics, differing background principles, and the influence of grid refinement. The results obtained with these three approaches have been compared with experimental data and numerical results available in the literature.
From the results presented one can draw the following conclusions:

- The numerical uncertainties for the wetted-flow calculations were small, the computational costs low and the iterative convergence optimal. Nevertheless, some differences between the numerical results and the reported experimental lift coefficient have been seen, which may indicate some discrepancies between the real experimental setup and the numerical setup here used, and by almost all authors that have addressed the same case.

- The grid design, utilising high-quality structured grids and locally nested refinement, increased the numerical resolution in the cavitation (and DDES) regions of interest. This is usually only done when employing unstructured grids.

- The numerical demands in terms of grids, time-step and non-linear iterations increase significantly in the case of cavitating-flow. For the RANS- Reboud correction case, these are even more significant. For the same numerical settings, DDES calculations delivered better iterative convergence at lower computational costs than the RANS- Reboud ones.

- The effect of grid refinement is different for all three methods used: 1) for RANS, the finer the grid the higher the values of eddy-viscosity and the worse the numerical predictions in
terms of cavity dynamics; 2) for DDES, the finer the grid the more turbulent structures are solved, the lower the eddy-viscosity levels, and the better the cavity dynamics; 3) for RANS-Reboud correction, refining the grid leads to non-monotonic convergence behaviour, which increases the numerical difficulties (robustness) of applying this correction.

- The cavity growth, shrinking and detachment was better captured using a RANS-Reboud correction approach than DDES. This is due to the shielding of the boundary layer from turbulence resolution intrinsic to this SRS model.

- The loads on the wing, and associated cavitation extents, were underpredicted when compared with the available experiments and some results of the literature.

In terms of future work, the authors would like to perform calculations for the same test case on finer grids, use the complete wing and study the effect of the time resolution. To employ other SRS methods like IDDES, XLES or PANS, or more advanced RANS turbulence models such as EARSM and RSM would also be the logical next step to improve the numerical predictions. Additionally, automatic grid and time-step adaptation (refinement and coarsening) would be a good technique to further reduce numerical dissipation, increase grid resolution in all vapour areas, and decrease the computational cost of these calculations.

ACKNOWLEDGMENTS

The authors would like to thank Maarten Kerkvliet and Bart Schuiling of MARIN for their help with the grid generation. This research is partially funded by the Dutch Ministry of Economic Affairs. This support is gratefully acknowledged.

References


NUMERICAL STUDY OF CAVITATION ON A NACA0015 HYDROFOIL: SOLUTION VERIFICATION

CARLO NEGRATO∗†, THOMAS LLOYD†, TOM VAN TERWISGA†, GUILHERME VAZ† AND RICKARD BENSWO∗

∗Department of Shipping and Marine Technology
Chalmers University of Technology
Campus Lindholmen, SE-41296, Gothenburg, Sweden
email: negrato@chalmers.se - Web page: http://www.chalmers.se/

† Research and Development Department, †MARIN Academy
Maritime Research Institute of The Netherlands (MARIN)
P.O. 28, 6700AA, Wageningen, The Netherlands
Web page: http://www.marin.nl/

Key words: Cavitation, Verification, NACA0015 foil, RANS, Discretization Error.

Abstract. The present paper analyses a series of Computational Fluid Dynamic simulations of the cavitating flow around a two-dimensional NACA0015 foil. The foil is placed at 6° angle of attack and the cavitation number is 1.1. Two mesh designs, namely a block-structured topology and an unstructured topology, are compared; additionally, grid refinements and time step refinements are carried out. Solution Verification is addressed with calculation of the discretization error and the numerical uncertainty. The numerical uncertainty for the average lift coefficient is found to be large, up to 15%. The reason is the difficulty of achieving a grid independent solution: with very fine meshes, the flow shifts from an attached, oscillating sheet cavity pattern to a regime dominated by shedding of cavity clouds. On the other hand, neither the time resolution nor the choice of grid topology influence largely the flow pattern; instead, they only lead to differences in the maximum and minimum cavity size.

1 INTRODUCTION

The objective of Computational Fluid Dynamics (CFD) tools is to find the solution for the mathematical equations of the model which describes the flow dynamics. The governing equations are solved numerically by aid of computational resources. The numerical approach becomes necessary when the analytic solution is either unknown or impossible to determine. However, the discrete approach leads, in fact, to an approximation of the mathematical equations, hence it brings errors into the solution. Additionally, sources of errors are introduced when iterative algorithms are used to overcome the non-linearities in the model. As a result, the solution obtained from CFD codes is subject to multiple error contributions. The activity which deals
with estimation of errors and uncertainties in a numerical simulation is referred to as Solution Verification\[1\]. Furthermore, for practical applications, CFD codes are used to predict complex flow patterns and often involves the modeling of additional physical phenomena. This is the case for applications where hydrodynamic cavitation occurs. Cavitation is the change of phase of a flowing liquid when the local pressure falls close to the vapor pressure. Marine propellers are one example where cavitation influences the performance of the system: it affects the propeller efficiency and it can lead to undesirable effects, such as an increased noise level, induced vibrations or surface erosion. For a numerical simulation of complex flows, it becomes necessary to estimate the numerical error and provide an estimate of the uncertainty in the results; Solution Verification not only is needed to support the credibility of the numerical results, but the uncertainty level is useful when CFD is used in design tasks.

The scope of the present work is to provide Solution Verification for the cavitating flow around a NACA0015 hydrofoil. This test case has been investigated earlier, both numerically\[2\],[\[3\] and experimentally\[4\],[\[5\] but Solution Verification for the numerical calculations was not tackled so far. Furthermore, several examples of Solution Verification exercises are found in literature, for instance in the cases of cylinder flows\[6\]. However, to our knowledge, verification has not been addressed for a test case with unsteady cavitating flow. The foil is placed in a water tunnel at an angle of attack of $6^\circ$. With the cavitation number equal to 1.1, the flow is unsteady. Therefore, to compute the numerical error/uncertainty, we rely on a method based on both grid and time step refinement. A series of RANS simulations is carried out with different grid densities and time steps. The large dataset allows a deeper analysis of the influence of spatial and temporal resolution. The work is complemented with a study of the effect of two different grid designs, with one set of structured grids and a second set of unstructured ones.

The paper is organized as follows: the first section explains the numerical background (§2.1), the details of the test case are given in §2.2 and a description of the method employed for calculation of errors/uncertainties is provided in §2.3. The following sections give the results of this study, with separate sub-sections regarding the numerical uncertainty (§3.1), the effect of grid design (§3.2), grid density (§3.3) and time step (§3.4). Finally, the conclusions and recommendations for future work are summarized in §4.

2 METHODOLOGY
2.1 Numerical model

The multi-phase flow is modeled using a mixture approach. The governing mass and momentum equations for an incompressible flow are solved considering the mixture density and the mixture viscosity; an additional transport equation for the vapor volume fraction $\alpha_v$ is added, which includes a source term to mimic the phase change. The source terms are computed by means of a modified Sauer model\[7\]. The turbulence modeling relies on the unsteady Reynolds-averaged-Navier-Stokes (RANS) equations, with turbulent viscosity assumption for the Reynolds stresses; all the simulations use the 2-equation $k-\sqrt{k}L$ model for turbulence closure\[8\].

The CFD viscous-flow code ReFRESCO (http://www.refresco.org/) is employed. ReFRESCO implements a finite volume, co-located discretization method, while a SIMPLE-type
algorithm is used to solve the coupled equations in a segregated manner. The convective flux in the momentum equation is discretized using a QUICK scheme. In view of the experience with simulation of cavitating flow with ReFRESCO\cite{9}, a first-order implicit Euler scheme is used for time integration. As a result, the spatial-temporal numerical scheme is of mixed-order type\cite{10}.  

2.2 Test case and computational setup  

The two-dimensional NACA0015 hydrofoil has a chord length $c = 0.2 \text{ m}$ and it is placed at $\alpha = 6^\circ$ angle of attack in a water tunnel of height equals to $2.85c$ (Figure 1). A nominal width $w$ of one chord is set. The flow domain extends $2c$ upstream of the leading edge and $4c$ downstream of the trailing edge. A pressure probe is located at the top boundary; the reference system is centered at the center of gravity of the foils (i.e. at a relative chordwise position of 0.3086). To reduce the computational cost, slip velocity is allowed at the tunnels walls. An inflow velocity $U_\infty = 6 \text{ m/s}$ results in a Reynolds number based on chord length $Re = 1.2 \cdot 10^6$, given the property of (liquid) water: $\rho_l = 998 \text{ kg/m}^3$, $\mu_l = 1.002 \text{ kg/(ms)}$. The vapor density is set to $\rho_v = 0.024 \text{ kg/m}^3$. The cavitation number is $\sigma = 2(p_{ref} - p_v)/\rho U_\infty^2 = 1.1$. The test case has been widely investigated experimentally and numerically, among others by Arndt\cite{4} and Hoekstra\cite{2}. The latter reports the value of $\sigma = 1.1$ as the cavitation number at which the transition occurs between a regime where an attached sheet cavity develops and a regime where shedding of cavitation clouds takes place.

Two sets of five grids each were generated. The meshing tools GridPro and Hexpress were used to create a set of block-structured grids and a set of unstructured grids respectively. The grids were designed to ensure a good resolution close to the suction side of the foil, where cavitation develops. Figure 2 gives a view of the two coarsest grids in the sets. For the unstructured grids, the cells are aligned with the undisturbed flow direction, with the exception of the boundary layer region; the coarsening towards the far field is sharper than the block-structured grid. Furthermore, the sets of structured grids are geometrical similar, which means that the grid properties (skewness, orthogonality, etc.) do not change with the grid refinement and the refinement ratio is constant over the domain. Differently, for the unstructured meshes geometrical similarity is obtained in the boundary layer cells as well as far from the foil, but in the transition region between the wall layers and the outer grid similarity is not guaranteed. All the grids were designed to fully resolve the boundary layer; the maximum value of $y^+$ is found at $y_{max}^{+} = 2.4$ and $y_{max}^- = 2.2$ for the coarse structured and coarse unstructured meshes respectively.
Table 1: Number of cells $N$ and grid refinement ratios $h_i/h_1$ for the two grid sets used. Time steps $t_i$ and time step refinement ratios $t_i/t_1$.

<table>
<thead>
<tr>
<th>Case ID</th>
<th>Structured GridPro</th>
<th>Unstructured Hexpress</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>GP5</td>
<td>GP4</td>
</tr>
<tr>
<td>$N \times 10^{-3}$</td>
<td>29.6</td>
<td>66.3</td>
</tr>
<tr>
<td>$h_i/h_1$</td>
<td>3.97</td>
<td>2.66</td>
</tr>
<tr>
<td>Time steps ID</td>
<td>T4</td>
<td>T3</td>
</tr>
<tr>
<td>$t_i [s]$</td>
<td>$2.667 \times 10^{-4}$</td>
<td>$1.333 \times 10^{-4}$</td>
</tr>
<tr>
<td>$t_i/t_1$</td>
<td>8.00</td>
<td>4.00</td>
</tr>
</tbody>
</table>

Figure 2: Overview of the coarse block-structured mesh and the coarse unstructured mesh.

Table 1 shows the number of cells and the grid refinement ratio $h_i/h_1$ which, for a two dimensional mesh, is defined as

$$\frac{h_i}{h_1} = \sqrt{\frac{(N_{\text{cells}})_1}{(N_{\text{cells}})_i}}, \quad (1)$$

where $h_i$ is the typical cell size. $h_1$ refers to the finest mesh in the set. Efforts were put into producing grids with a comparable number of cells and similar refinement ratios between the structured and unstructured sets. Lastly, four time steps are used in combination with all the grids. The time step refinement ratio is simply the ratio of time step $t_i$ to the finest $t_1$.

2.3 Solution verification

An extensive explanation of Verification & Validation tasks for CFD simulations is given by Roache\cite{1} and summarized by Eça\cite{11}: the scope of verification is to show that we are “solving the equations right”, while the scope of Validation is to show that we are “solving the right equations”. Following this distinction, verification provides the numerical error/uncertainties while Validation deals with the modeling error/uncertainties.

In this work, Solution Verification is provided using the method of Eça and Hoekstra\cite{11}. One example of application of this method for unsteady flows is given by Rosetti\cite{6}. The numerical error is commonly split into three contributions: the round-off error, the iterative error and
the discretization error. The round-off error is a consequence of the finite machine precision. The iterative error relates to the iterative methods employed to solve the non-linearity of the governing equations. While the round-off error is reduced to negligible values by use of double precision (15 digits), the iterative error can become non-negligible for complex flows. Furthermore, the cavitating flow around the hydrofoil is unsteady, hence the iterative error affects the solution at each time step. The iterative error is estimated using the infinity norm $L_\infty(\phi)$ of the normalized residuals. According to Rosetti\cite{6}, a value of $L_\infty(\phi) < 10^{-6}$ for unsteady simulations will reduce the iterative error to a negligible level. However, for the test case investigated in this study, it was not always possible to converge the simulations to this level. Some large values of $L_\infty(u_x) \approx 10^{-3}$ for the axial velocity are seen for the simulations with the finest grid and the coarsest time steps. Because in the Eça and Hoekstra method it is assumed that the discretization error is dominant, the simulations which exhibit larger residuals are discarded. In the other cases, the infinity norm is converged to values between $10^{-8} < L_\infty < 10^{-4}$, depending on the flow variable and the simulation time.

The method uses a power expansion to compute the discretization error $e(\phi_i)$ for the quantity of interest $\phi_i$ for each case of grid and time step\cite{11}. There are ten error estimators featured in the Eça and Hoekstra method, which make it suitable for practical application where some scatter in the data might occur. The error estimator provides a fit through the datapoint and allows to extrapolate the exact solution. From the knowledge of the discretization error, the uncertainty is computed, which is defined as the range that contains the exact solution within a 95\% accuracy. To compute the uncertainty, the method takes three factors into account: the standard deviation of the fit, the difference between the actual data point and the value obtained through the fit and a variable safety factor.

3 RESULTS

The cavitating flow over the NACA0015 was simulated with all combinations of the grids and the time steps of Table 1. As it will be shown in Section §3.3, the flow behavior depends largely on grid refinement. The flow is characterized by a sheet cavity developing at the suction side of the foil; depending on the numerical settings, the cavity either oscillates (with a periodic growth and shrinking), or it breaks up to shed a cloud which travels downstream.

In the analysis of the results, the outcome of the numerical uncertainty analysis is given in the first section. Then, the influence of spatial and temporal discretization is addressed in more detail; for the purpose, the simulation with a structured grid of approximately $118 \times 10^3$ cells and a time step of $1.333 \times 10^{-4}$ (i.e. case GP3-T3) is taken as a reference.

3.1 Discretization uncertainty

The estimation of the numerical uncertainty is made for the average lift coefficient $C_L = 2L/(\rho U_\infty^2 c w)$. It is computed from the time history of the lift force in the last 12 cycles. Because of the aforementioned weaker iterative convergence for the cases with the finest grid and the coarsest time step, the solutions from the cases GP1-T4 and GH1-T4 are discarded in the error estimation.

Figure 3 shows the surface fit used to extrapolate the exact solution. Furthermore, Table 2 provides the extrapolated exact solution $C_{L_0}$, the error for the finest resolution $e(C_{L_1})$, the
Figure 3: Values of lift coefficient from the datasets (black spheres), surface fits used for error estimation and uncertainty range (green error bars). Left: structured grid dataset. Right: unstructured grid dataset.

Table 2: Extrapolated exact solution, discretization error, uncertainty and orders of convergence. (1, 2) indicates that a first-plus-second order fit is used, as featured in the Eca and Hoekstra method[11].

<table>
<thead>
<tr>
<th>Structured GridPro</th>
<th>Unstructured Hexpress</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_{L_0}$</td>
<td>$C_{L_1}$</td>
</tr>
<tr>
<td>0.564</td>
<td>0.549</td>
</tr>
</tbody>
</table>

numerical uncertainty and the orders of convergence. Although the percentage error for the finest simulations is <3% for both grid topologies, the uncertainty is rather large: 15% for the structured grids and 8.4% for the unstructured grid. Furthermore, the formal order of convergence (second order in space and first order in time) is not retrieved for neither dataset, which is a result of the scatter seen in the datasets.

With the fine grids, shedding of cavity clouds occurs, which does not happen with coarser grids. This additional dynamic phenomena leads to major changes on the forces exerted on the foil. Furthermore, when shedding occurs, the signal for the lift coefficient is not fully periodic, which also induces some disturbances when computing its average. The shift from a fully attached sheet to a shedding cavity regime depending on grid refinement level leads to scatter in the datasets, hence the large uncertainty values follow.

3.2 Influence of grid topology

In this section, a comparison of the reference simulation (GP3-T3) with structured grid and the simulation with unstructured grid (GH3-T3) is provided. The two cases have the same time step $t_3$ and the number of cells is comparable. Hence, the influence of two different topologies is
addressed. The analysis of the results is focused on the integral quantities (Table 3). Also, we look at the time traces and provide a frequency analysis (Figure 4) for three selected quantities: the total vapor volume, the lift coefficient and the pressure at the top tunnel probe. The pressure coefficient is defined as \( C_p = 2p/(\rho U_\infty^2) \).

**Table 3**: Drag coefficient, lift coefficient and Strouhal number. Relative differences (\( \Delta \)) between the results with a structured and an unstructured grid topology are included.

<table>
<thead>
<tr>
<th>Case</th>
<th>( \bar{C}_L )</th>
<th>( \Delta \bar{C}_L ) (%)</th>
<th>( \bar{C}_D )</th>
<th>( \Delta \bar{C}_D ) (%)</th>
<th>( \bar{C}<em>{p</em>{\text{min}}} )</th>
<th>( \Delta \bar{C}<em>{p</em>{\text{min}}} ) (%)</th>
<th>( St )</th>
<th>( \Delta St ) (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>GP3-T3</td>
<td>0.520</td>
<td>0.0375</td>
<td>0.0358</td>
<td>4.52</td>
<td>-1.231</td>
<td>1.84</td>
<td>0.115</td>
<td>0.115</td>
</tr>
<tr>
<td>GH3-T3</td>
<td>0.518</td>
<td>0.0358</td>
<td>0.0358</td>
<td>4.52</td>
<td>-1.253</td>
<td>1.84</td>
<td>0.115</td>
<td>-</td>
</tr>
</tbody>
</table>

**Figure 4**: Time traces and harmonic contents of the total vapor volume, lift coefficient and pressure at the probe. Dashed: reference simulation GP3T3 with structured grid (\( \approx 118k \) cells). Solid: unstructured grid topology, case GH3T3 (\( \approx 171k \) cells). The time step is \( t_3 = 1.333 \times 10^{-4} \) s for both topologies.

Finally, the Strouhal number is based on the upstream velocity and the chord length, and it is computed using the signal of the lift coefficient. The flow pattern is dominated by the growth and shrinking of an attached sheet cavity. Both topologies result in regular oscillations. The average lift coefficient differs by less than 1%, but the difference in drag coefficient is larger (4.52 \%). Although the predicted frequency of cavity oscillation coincide (\( St = 0.115 \)), the sheet cavity is longer for the structured grid during most of the cycle. Hence, the total vapor volume is larger for the structured grid, as visible in the time traces of Figure 4. However, despite the different cavity size, the instantaneous lift differs by 4% at maximum between topologies. A deeper analysis of the flow fields shows that a stronger re-entrant jet develops for the unstructured grid. The low pressure in the region of (liquid) flow where the re-entrant jet develops, partially balances the smaller cavity extension. Finally, the regular, periodic cavity oscillation is reflected in the frequency contents, where the first harmonic component is dominant for all three signals.
3.3 Influence of grid size

Secondly, the influence of grid size is investigated by a comparison between the reference solution and the solution with the finest time step in the set, i.e. $t_1 = 3.333 \times 10^{-5}$. Table 4 reports the integral values while Figure 5 gives the time histories. Furthermore, Figure 6 shows a side-by-side comparison of pressure contour plots at selected time instants. With a medium grid the attached cavity oscillates regularly between a minimum length and a maximum length, shown in the top left and bottom left plots. Remarkably, with a fine grid the predicted cavitation dynamics changes radically: a vapor structure is detached from the rear edge of the sheet cavity, as a consequence of a re-entrant jet flow which breaks up the sheet cavity. The cavity bubble is convected downstream, as shown in the two right contour plots. It moves towards the trailing edge, where it interacts with the trailing edge flow and eventually collapses downstream of the foil. Moreover, when the shed vapor structure is small, it can collapse before reaching the trailing edge. The highly dynamic cavitation cycle results in large variations of total vapor volume and lift coefficients among cycles, visible in the first time trace of Figure 5.

The largely different dynamic behavior predicted with a fine grid motivates the large differences in average loading coefficients (Table 4). The average lift coefficient differs by 4% between grids and the average drag coefficient even by 31%!

When looking at the time history, it is seen that the signals for GP1 (solid lines in Figure 5) do not show a clear period behavior. Nevertheless, the harmonic content is dominated by a first harmonic component for both the lift force and the pressure at the probe. Differently, for the total vapor volume there is no peak at the shedding frequency, but rather a broad band content in between the first harmonic $n = 1$ and the second harmonic $n = 2$.

Additionally, Figure 7 shows the average pressure coefficient distribution on the surface of the foil as well as its standard deviation. The standard deviation gives a measure of the influence of the dynamic cavitation cycle on the surface pressure. At the pressure side, the average pressure does not differ between grids. At the suction side, $C_p$ is equal to $-\sigma$ between $-0.2 < x/c < 0.1$ for the medium grid. This is the region where the sheet cavity is continuously seen. Correspondingly, the standard deviation is zero. Furthermore, there is a quick pressure recovery in the average $C_p$ for $x/c > 0.1$. The pressure recovery is milder for the fine grid solution because of the decrease in surface pressure induced by the cavity bubble traveling downstream. Finally, the right plot shows that the standard deviation is larger for the fine grid at both the suction side and the pressure side; the increase towards the trailing edge comes as a consequence of the interaction of the traveling bubble with the trailing edge flow. The peak is located at $x/c = 0.1$ and $x/c = 0.15$ for the medium and fine grid respectively. Because the surface pressure within the cavity is constant and equal to the vapor pressure, these chordwise coordinates represent

Table 4: Drag coefficient, lift coefficient and Strouhal number. Relative differences ($\Delta\ast$) between the results with a medium grid GP3 and a fine grid GP1 are included.

<table>
<thead>
<tr>
<th>Case</th>
<th>$C_L$</th>
<th>$\Delta C_L$(%)</th>
<th>$C_D$</th>
<th>$\Delta C_D$(%)</th>
<th>$C_{p_{min}}$</th>
<th>$\Delta C_{p_{min}}$(%)</th>
<th>$St$</th>
<th>$\Delta St$(%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>GP3-T3</td>
<td>0.520</td>
<td>3.58</td>
<td>0.0375</td>
<td>31.0</td>
<td>-1.231</td>
<td>2.84</td>
<td>0.115</td>
<td>9.10</td>
</tr>
<tr>
<td>GP1-T3</td>
<td>0.539</td>
<td>3.88</td>
<td>0.0544</td>
<td>30.0</td>
<td>-1.197</td>
<td>2.84</td>
<td>0.104</td>
<td>9.10</td>
</tr>
</tbody>
</table>

164
Figure 5: Time traces and harmonic contents of the total vapor volume, lift coefficient and pressure at the probe. Dashed: reference simulation with medium grid GP3 ($\approx 118k$ cells). Solid: results with fine grid GP1 ($\approx 468k$ cells). The time step is $t_3 = 1.333 \times 10^{-4}$ s for both grids.

Figure 6: Contour plots of pressure coefficient. Isolines of $C_p = -\sigma = -1.1$ (green) and $\alpha_v = 0.5$ (black). Left column: reference simulation, snapshots at minimum and maximum cavity extension. Right column: snapshots with fine grid, showing the shed cavity traveling downstream during a period of time of $\approx \frac{4}{3}T_S$.

Figure 7: Average pressure coefficient distribution on the foil (left) and standard deviation (right). Dashed: reference simulation with medium grid GP3. Solid: results with fine grid GP1.
the points of maximum cavity length for the two simulations.

The change in cavitation dynamics with a fine grid is seen also with the finer time steps $t_2, t_1$. Besides, the same conclusions are drawn when looking at the results with the unstructured Hexpress grid topology, although the shedding behavior for the unstructured grid is less dynamic, with a more upstream and less violent collapse of the shed cavity.

3.4 Influence of time step

A third sensitivity study is carried out, regarding the effect of a change in time step while keeping the same grid resolution. The reference solution GP3-T3 with a time step $t_3 = 1.333 \times 10^{-4}$ s is compared to the solution GP3-T1 with a time step $t_1 = 3.333 \times 10^{-5}$ s. The outcome is given in Table 5 and Figure 8. Dimensionful time is used for the time traces, hence the phase shift seen is due solely to the initial, transient, part of the simulations (not shown here).

With both time steps, an attached sheet cavity remains, without shedding. The shape of the cavity is the same for the two solutions; however, with a smaller time step the attached cavity is both longer and thicker at its maximum length, which results in larger peaks in total vapor volume. In addition, a fine time step results in a smaller minimum cavity length. Correspondingly, the oscillations in lift coefficients and pressure at the top tunnel wall have larger amplitudes. Nevertheless, the difference in shedding frequency (hence the Strouhal number) is negligible.

Moreover, the frequency content of Figure 8 has a visible larger harmonic components for the solution with small time step, mostly in the second harmonic. This contribution is related to the behavior of the cavity at its early stage, when the cavity is at its minimum length: before the attached sheet cavity starts to grow again, a rapid oscillation occurs. The time scale of such effect is on the order of $10^{-2}$ s, hence two to three orders of magnitude larger than the time steps. Nonetheless, only the simulation with fine time step shows this behavior.

4 CONCLUSIONS

The present work tackles the Solution Verification for the cavitating flow over a two-dimensional NACA0015 hydrofoil placed in a water tunnel at an angle of attack of 6°. The cavitation number was $\sigma = 1.1$. Multi-phase, viscous flow simulations were run using two grid topologies: a block-structured topology and a fully unstructured one. Furthermore, combinations of five systematically refined grids and four time steps were considered. Hence, it was possible to investigate the influence of discretization levels in time and space.

Discretization error and numerical uncertainty are computed. The numerical uncertainty for the average lift coefficient was found to be as large as 15% and 8.4% for the finest structured and unstructured grid respectively. The large value of the uncertainties is a consequence of scatter

Table 5: Drag coefficient, lift coefficient and Strouhal number. Relative differences ($\Delta \ast$) between the results with a reference time step $t_3$ and a fine time step $t_1$.

<table>
<thead>
<tr>
<th>Case</th>
<th>$\overline{C_L}$</th>
<th>$\Delta \overline{C_L}$ (%)</th>
<th>$\overline{C_D}$</th>
<th>$\Delta \overline{C_D}$ (%)</th>
<th>$\overline{C_{p_{min}}}$</th>
<th>$\Delta \overline{C_{p_{min}}}$ (%)</th>
<th>$St$</th>
<th>$\Delta St$ (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>GP3-T3</td>
<td>0.520</td>
<td>3.01</td>
<td>0.0375</td>
<td>13.4</td>
<td>-1.231</td>
<td>-0.05</td>
<td>0.115</td>
<td>0.01</td>
</tr>
<tr>
<td>GP3-T1</td>
<td>0.536</td>
<td>3.01</td>
<td>0.0433</td>
<td>13.4</td>
<td>-1.232</td>
<td>0.05</td>
<td>0.115</td>
<td>0.01</td>
</tr>
</tbody>
</table>
in the results, due to a shift from an attached sheet cavity flow to a shedding cavity behavior for the finest spatial resolutions.

It was observed that the grid resolution has the largest influence on the flow dynamics. With a fine grid (and a medium-to-fine time step) cavity clouds are shed from the rear edge of the cavity. The clouds are convected downstream and collapses either before reaching the trailing edge or in the wake of the foil, depending on the size of the bubble. The test case is two-dimensional, and the flow modeling relies on RANS for turbulence and mixture approach for multi-phase flow. Furthermore, for the sake of the verification study, the grid densities used are rather fine, compared to what is commonly seen for unsteady RANS simulations with the same setup. Bearing in mind the assumption of the test case, the numerical settings and the level of flow modeling, it is concluded that the strive towards a grid independent solution turns out to be vain: very fine spatial and temporal resolutions leads to additional cavitation dynamics.

Finally, both the grid topology and the temporal discretization are found to provide small effects, in comparison to the large influence of grid density. With the unstructured Hexpress grids the attached sheet cavity is shorter and thinner, but the same periodic behavior of the structured GridPro grids is observed. Regarding the influence of temporal discretization, a smaller time step leads to larger oscillations of the attached sheet cavity. In addition, a small contribution from the second harmonic appears, as a result of rapid oscillation of the cavity at the beginning of the cycle, when the cavity has the minimum length. Future work is recommended to extend the current study. For the fine grids, the lack of clear periodicity is expected to affect the computation of mean values. A more thorough analysis, possibly including calculation of the statistical uncertainty, is suggested. Moreover, Solution Verification is naturally followed by Validation. In this perspective, extension to a three dimensional foil is recommended to have a more consistent comparison with the experiments.
ACKNOWLEDGEMENTS

This study was funded partly within the H2020 project LeanShips (grant no 636146) and partly within MARIN Academy funds. Simulations were performed on resources at Chalmers Centre for Computational Science and Engineering, C3SE, provided by the Swedish National Infrastructure for Computing (SNIC). The authors also wish to thank Maarten Kerkvliet (MARIN) for providing the GridPro grids.

REFERENCES

EFFECTS OF BLADE GEOMETRY ON CAVITATION AND PRESSURE FLUCTUATIONS OF TUNNEL THRUSTERS

CHENG YU, XIAO-QIAN DONG*, WEI LI, CHEN-JUN YANG

State Key Laboratory of Ocean Engineering(SKLOE)
Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration(CISSE)
Shanghai Jiao Tong University, Shanghai 200240, China
*Email: xiaoqiandong0330@sjtu.edu.cn

Key words: tunnel thruster, model test, CFD, blade geometry, cavitation, fluctuating pressure

ABSTRACT: Compared with open propellers, tunnel thruster blades are more vulnerable to cavitation and local structure vibration problems because they are typically heavily loaded and subject to severe non-uniformity of inflow produced by the blunt gearbox. However, it seems that the simple 'flat plate' is still often used in designing the thruster blades. In this research, model tests and RANS simulations are carried out for three highly skewed thruster blades having different pitch and rake profiles to investigate the effects of blade geometry on cavitation and pressure fluctuations. The results indicate that the 'flat plate' blade is unfavorable for vibration excitation and unloading towards the tip is an effective way to reduce the fluctuating pressures.

1 INTRODUCTION

Ship vibration can bring about structural damage, fatigue, excessive noise and other issues. The major sources of excitation include the propeller, main engine, and waves, where the propeller is usually the most important one. Once cavitation happens, the propeller induced fluctuating pressures will increase significantly and such problems are among the most active research topics in ship propulsion.

For open propellers, relevant researches based on both model experiments and numerical simulations are relatively sufficient. For example, Pereira et al.[1,2] conducted fluctuating pressure and noise measurements for a cavitating propeller in uniform and non-uniform flows and demonstrated that the pressure fluctuations due to the occurrence of cavitation are proportional to the cavity volume acceleration by the test results. Salvatore et al.[3] compared seven computational models including RANS, LES, and BEM for the INSEAN E779A propeller in uniform and non-uniform inflows. The comparison of numerical results highlights a good agreement for the non-cavitating steady flow predictions, whereas for the cavitating flow, discrepancies in cavity extent are observed. The main reason is likely to be the lack of grid density and/or too much numerical dissipation in the vicinity of the cavity-fluid interface.

However, the relevant researches on tunnel thrusters seem to be scarcely available in the public domain due to high loading on the impeller blades and interactions among different parts of the thruster. The typical configuration of a tunnel thruster includes an impeller, a T-shaped...
housing (the 'gearbox' hereinafter) of the right-angle shaft system, and a driving motor. Due to
the limited space available, the gearbox is typically blunt in geometry and close to the impeller
blades, which induces severe blockage effect and flow non-uniformity for the impeller.
Meanwhile, the impeller blades are usually heavily loaded. Under such adverse conditions
tunnel thrusters are more vulnerable to cavitation, especially when the simple 'flat plate'
impeller blades are used. The fluctuating pressures induced by the impeller on the tunnel wall
can exceed those by a propeller on the stern by two orders of magnitude when cavitation
happens.

There are just a few pieces of work, all published in the 1960s, focusing on the hydrodynamic
performance of tunnel thrusters based on model experiments\cite{4} and design methods\cite{5-7}. During
the last ten years advances in computational fluid dynamics have made it possible to simulate
the viscous flow of tunnel thrusters\cite{8-10}. Unfortunately, the research work on cavitation and its
induced effects is still scarce in the public domain. Stefano et al.\cite{11} investigated the
hydrodynamic performance and cavity patterns for a 'flat plate' Kaplan type propeller working
in a cylinder at two pitch settings, based on BEM simulation and experimental observation.
Fischer\cite{12} proposed a design criterion for tunnel thrusters from the perspective of reducing the
vibration and noise and measured the noise levels in cabins. It has been found that noise and
vibration levels are different depending on thrust direction and the noise is 5-10 dB higher in
low frequency range when the gearbox is located upstream of the impeller.

As dynamic positioning systems are equipped on more ships and operate more frequently,
the necessity becomes obvious to enhance the performance of thrusters. In this research, model
tests and RANS simulations are carried out for three highly skewed thruster blades having
different pitch and rake profiles to investigate the effects of blade geometry on cavitation and
pressure fluctuations. The impeller models are designed to produce the same amount of thrust
when fitted to the same gearbox and bow model. The fluctuating pressures on the tunnel wall
are measured at a number of locations in the vicinity of the blade tip. Viscous flow CFD
simulations are carried out for the three impellers in one condition to gain more detailed
information of the flow.

2 EXPERIMENTAL RESEARCH

2.1 Test facility and measuring equipments

The model tests are carried out in the cavitation tunnel of Shanghai Jiao Tong University, as
shown in Figure 1. The test section is 6.1m in length, and its cross section is 1m×1m with
rounded corners. The axial flow velocity over the test section ranges from 0.5m/s to 15.8m/s,
and the static pressure at the centerline of the test section ranges from 25kPa to 300kPa. The
non-uniformity of axial flow velocity is less than 1%.

As shown in Figure 2, the generic bow model is 0.58m long, with identical cross section
geometry over the length and a tunnel in transverse direction to house the thruster. The bow
model is made of plexiglass to facilitate observation of the flow and cavitation inside the tunnel.
As illustrated in Figure 3, the bow model is installed at the streamwise center of the third
observation window, and at a height to ensure the impeller shaft axis is 0.5m above the bottom floor of the test section.

**Figure 1:** The cavitation tunnel of Shanghai Jiao Tong University

**Figure 2:** The generic bow model  
**Figure 3:** Setup of the bow model

As illustrated in Figure 4, six pressure transducers are installed on the tunnel wall, with the measuring surfaces flush with the inner wall surface. They are numbered as S1 through S6. Transducers S2 through S5 are located in the plane perpendicular to the shaft axis, and passing the mid-chord point of the tip section. Circumferentially, the four transducers are arranged symmetrically port and starboard at a spacing of 12°. Transducers S1 and S6 are arranged at 12 o'clock position, 0.127*D upstream and downstream of the plane where S2 through S5 are arranged. Here D denotes the impeller diameter.

**Figure 4:** Arrangement of pressure transducers
The fluctuating pressures are measured with CYG505AFM micro pressure transducers made by Kunshan-ShuangQiao, a company in China. The measurable pressure range is 0–800kPa, and the accuracy and hysteresis repeatability are 0.5% and 0.05%, respectively. For the acquisition and analysis of the pressure signals, the PXIe-4331 multi-channel synchronous signal acquisition instrument made by National Instrument is utilized. The gain error is 0.1% of the reading and the offset error is 198μV/Vex. Figure 5 shows the equipments for pressure measurements.

![Figure 5: Equipments for fluctuating pressure measurement](image)

### 2.2 Impeller models

Three impeller models, named P1, P2, and P3 respectively, are utilized for the present research to investigate the effects of pitch distribution and local rake near the tip of blade. They were designed to produce the same amount of thrust at the same speed of rotation. According to our RANS simulations, the thrust coefficients of impellers P1, P2, and P3 are 0.183, 0.182, and 0.183, respectively.

<table>
<thead>
<tr>
<th>Impeller model</th>
<th>P1</th>
<th>P2</th>
<th>P3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Direction of rotation</td>
<td>Left-handed</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Number of blades</td>
<td>4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Diameter (mm)</td>
<td>250</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hub ratio</td>
<td>0.4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Expanded area ratio</td>
<td>0.6</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pitch ratio at 0.7R</td>
<td>0.75</td>
<td>0.89</td>
<td>0.89</td>
</tr>
<tr>
<td>Tip skew angle (deg.)</td>
<td>22.8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tip rake (mm)</td>
<td>0</td>
<td>-7.3</td>
<td>-7.3</td>
</tr>
<tr>
<td>Blade section</td>
<td>Symmetrical</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tip clearance (mm)</td>
<td>1.6</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The main particulars of the three impellers are listed in Table 1, where $R$ is the impeller tip radius, $R=D/2$. Impeller P1 is the traditional 'flat plate' design, *i.e.*, the blade mean
surface is a flat one (rather than a helical surface). The blades of P1 are rotated around their respective generator lines by 18.83° (from zero pitch). Impellers P2 and P3 were designed like a fixed pitch propeller with unloaded tips to improve cavitation performance. As shown in Figure 6, P2 and P3 share the same pitch distribution, which is quite different from that of P1. The three impellers share the same skew distribution, as shown in Figure 7. Impeller P2 differs from P3 only in total rake distribution, as shown in Figure 8. The former has a linear rake, while the latter has a sudden change in rake at 0.9\(R\) so that the tip region is bent towards upstream to alleviate the tip leakage vortex flow. Figure 9 shows the test models which are made from aluminum alloy with the surfaces anodized.

![Figure 6: Comparison of pitch distributions](image)

![Figure 7: Skew distribution of impeller models](image)

![Figure 8: Rake distributions of impeller P2 and P3](image)

![Figure 9: The impeller models for fluctuating pressure tests](image)

### 2.3 Test results and analysis

#### 2.3.1 Pre-test on the influences of flow speed and impeller rotation speed

The influences of flow speed and impeller rotation speed are investigated using impeller model P1. Tests are carried out at non-cavitating condition, where the absolute pressure of test section is 175kPa. The test conditions are listed in Table 2. For condition A and B, the impeller model works in 'quasi-bollard' condition, i.e., the test section inlet flow velocity, \(V\), is induced only by the tunnel thruster model. In condition C, the impeller of the cavitation tunnel rotates at a low speed, the inlet velocity is induced by both the tunnel thruster model and the impeller...
of the cavitation tunnel.

Table 2: Test conditions for investigating influences of impeller and inflow speeds

<table>
<thead>
<tr>
<th>Test condition</th>
<th>(P_n) (kPa)</th>
<th>(n) (r/s)</th>
<th>(V) (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>175</td>
<td>18</td>
<td>0.438</td>
</tr>
<tr>
<td>B</td>
<td>175</td>
<td>22.5</td>
<td>0.546</td>
</tr>
<tr>
<td>C</td>
<td>175</td>
<td>18</td>
<td>1.022</td>
</tr>
</tbody>
</table>

The fluctuating pressure coefficient is defined as

\[
C_p = \frac{p - \bar{p}}{\frac{1}{2} \rho n^2 D^2}
\]

where \(p\) is the measured absolute pressure, and \(\bar{p}\) is the time average of \(p\).

In Figure 10, a comparison is made for the total amplitude of fluctuating pressures coefficient, \(\tilde{C}_p\), which is defined as

\[
\tilde{C}_p = \sqrt{\sum_{i=1}^{5} i^2 \tilde{C}^2_{pi}}
\]

where \(\tilde{C}_{pi}\) denotes the \(i^{th}\) blade frequency harmonic component of fluctuating pressure coefficient obtained from Fourier analysis. The results in Figure 10 indicate that, within the range of test conditions shown in Table 2, the influences of flow speed and propeller rotation speed are almost negligible.

Figure 10: The influence of flow speed and propeller rotation speed on the total magnitude of fluctuating pressures for impeller model P1

2.3.2 Effects of blade geometry on pressure fluctuations

The tests on blade geometry effects are carried out in the 'quasi-bollard' condition. The rate of revolution, \(n\), is set to be \(22.5\) r/s based on considerations of the Reynolds number and the processing of fluctuating pressure data. At this rotating speed, the test section inlet flow velocities induced by the impeller model are \(0.55\) m/s, \(0.51\) m/s, and \(0.51\) m/s for P1, P2, and P3 respectively. Based on the induced flow and impeller rotation speeds, the Reynolds numbers at \(0.7R\) are equal to \(3.6 \times 10^6\) for the three impellers. For each impeller, the fluctuating pressures...
are measured at three conditions, $\sigma_n$=10.86, 3.75, and 2.5. The cavitation number, $\sigma_n$, is defined as

$$\sigma_n = \frac{p_0 - p_v}{\frac{1}{2} \rho n^2 D^2}$$

where $p_0$ is the static pressure at the shaft axis, $p_v$ is the vapor pressure of water, and $\rho$ is the density of water.

- **Cavity patterns**

![Cavity patterns](image)

Figure 11 shows the cavity patterns on the three impellers in different cavitation conditions. No cavity is observed at $\sigma_n$=10.86. At $\sigma_n$=3.75, the tip vortex cavity is already quite strong on impeller model P1, however, no tip vortex cavity is observed on impeller models P2 and P3, but some sheet cavity is observed instead. As the cavitation number decreases further to 2.5, for impeller P1, cavitation begins earlier in both radial and chordwise directions and develops into a sheet cavity, and the tip vortex cavity also becomes stronger. At the same cavitation
number, $\sigma_n=2.5$, the sheet cavity also becomes more extensive on impeller models P2 and P3, however the tip vortex cavities are still not obvious. The cavity patterns indicate that reducing the loading and using the nonlinear rake towards the tip, which is the case with P2 and P3 (see Figure 6 and Figure 8), can suppress the tip vortex cavitation, although the sheet cavitation in mid-radius region can become more extensive due to higher loading in that region.

- **Fluctuating pressures**

  Figure 12 shows the fluctuating pressure amplitudes at transducer S1, where the 1st blade frequency harmonic dominates. In non-cavitating condition, impeller P3 is clearly superior to P2 and P1 in terms of the pressure amplitude; however, when cavitation exists, both P2 and P3 are better than P1, and P2 is the best. At transducer S6, the downstream one, the pressure amplitudes of P3 are mostly the lowest compared with P2 and P1, as shown in Figure 13.

![Figure 12: Comparison of fluctuating pressure amplitudes at transducer S1 in non-cavitating and cavitating conditions](image1)

![Figure 13: Comparison of fluctuating pressure amplitudes at transducer S6 in non-cavitating and cavitating conditions](image2)
Figure 14 through Figure 16 compare the fluctuating pressure amplitudes of the three impellers at transducers S2 through S5 in non-cavitating and cavitating conditions. For impeller P1, the fluctuating pressure amplitudes increase significantly as the cavitation number decreases. However, this tendency is not obvious for P2 and P3. In general, the pressure amplitudes of P1 are higher than those of P2 and P3, especially at the 1st and 2nd blade frequency harmonics.

**Figure 14**: Comparison of fluctuating pressure amplitudes at transducers S2 through S5 in non-cavitating condition, $\sigma_n=10.86$.

**Figure 15**: Comparison of fluctuating pressure amplitudes at transducers S2 through S5 in cavitating condition, $\sigma_n=3.75$. 
Figure 16: Comparison of fluctuating pressure amplitudes at transducers S2 through S5 in caviting condition, $\sigma_n=2.5$

Figure 17 shows the comparison of the total amplitude of fluctuating pressures at the six transducer locations. The result is clear that the fluctuating pressure amplitudes of impeller P1 are higher than those of P2 and P3 in both caviting and non-caviting conditions and at all the measuring locations, which suggests that unloading the impeller blade towards the tip is an effective way to reduce the fluctuating pressures on the tunnel wall. In terms of the total amplitudes P2 and P3 are very close to each other, with only one exception at transducer S4, $\sigma_n=2.5$. In Figure 16, the fluctuating pressure amplitudes on transducer S4 are quite different from those on other transducers at the 1st and 2nd blade frequency harmonics, but the reason is not clear. Further study is necessary to find out if the rake distribution close to the tip can be further improved.

Figure 17: Comparison of the total amplitudes of fluctuating pressures
3 NUMERICAL SIMULATION

3.1 Modeling approach

Numerical simulations are conducted for the three impellers at $\sigma_n=2.5$, at rotating speed $n=22.5\, \text{r/s}$ by using the commercial CFD software FLUENT.

The geometric model of the tunnel thruster is illustrated in the left part of Figure 18 together with the bow model for the present numerical computations. The hull is defined in a fixed rectangular coordinate system, O-XYZ, where the X axis points toward downstream, the Y axis points vertically upwards, and the Z axis completes the right-handed system. The origin, O, is located at the center of the impeller. The impeller rotates about the X axis in its positive direction by the right-hand rule. The computational domain is a cuboid with the size of $120D \times 80D \times 40D$, as illustrated in the right part of Figure 18. Such large domain is necessary to ensure the computational stability.

![Figure 18](image)

The computational domain is divided into two parts. The first part contains the impeller blades and the hub, which is defined in a coordinate system rotating synchronously with the impeller. As shown in Figure 19, the rotating part is discretized with tetrahedral cells. The average value of $y^+$ on blade surfaces is about 100, and the maximum skewness coefficient of the cells is 0.79.

![Figure 19](image)

The rest of the computational domain is defined in the fixed coordinate system, and is further divided into 80 sub-domains which are discretized with hexahedral cells. The cells near the tunnel wall, especially around the entrance and exit of the tunnel are densified and stretched to
resolve the large gradients of flow quantities, as shown in 20.

The grids are generated by using GAMBIT, a pre-processor of the software FLUENT. There are about 10 million cells for the entire domain, where 6 million are dedicated to the rotating part.

The flow around tunnel thrusters is simulated by solving the RANS equations, using the shear-stress transport (SST) \( k-\omega \) model and Schneer & Sauer cavitation model for turbulence closure and cavitation simulation respectively. And for all the governing equations, the second-order upwind scheme is employed to discretize the convection terms.

The quasi-steady model is employed at first to improve the convergency of cavitation flow, then the unsteady sliding mesh model is applied to simulate the interactions between the rotating and stationary parts of the computational domain. The time step size is set as \( 2.469 \times 10^{-4} \) s, which corresponds to an angular displacement of 2° for the impeller rotating speed of 22.5 r/s. The environment pressure is set as \( 4.3 \times 10^3 \) Pa to meet with the requirement of \( \sigma_s = 2.5 \).

The surfaces of impeller blades, the hub, the gearbox, the tunnel, and the hull are set as stationary walls in their respective coordinate systems. The outlet is set as a pressure outlet, and the inlet and other side faces as designated in Figure 18 (right) are set as velocity inlets. Only the bollard condition is simulated here, so the inlet velocity is set to zero. Since the experiment is conducted in the cavitation tunnel, a symmetry boundary condition is employed for the top of the computational domain instead of the free surface.

3.2 Numerical results and analysis

- Cavity patterns

\[ \text{Figure 21: Comparison of cavity patterns for the three impeller models} \]

As shown in Figure 21, the simulated cavitation patterns at \( \sigma_s = 2.5 \) are quite similar to those observed in the experiments. The cavitation is reduced near the blade tip for impeller models P2 and P3 as a result of the tip-unloaded pitch distribution.
- Fluctuating pressures

The Fourier analysis results of the CFD simulated fluctuating pressures are compared with experimental results in Figure 22 ~ Figure 24. Figure 22 and Figure 24 indicate that the fluctuating pressure amplitudes at transducer S1 and S6 are well simulated.

**Figure 22**: Comparison of fluctuating pressure amplitudes at transducer S1 and S6, $\sigma = 2.5$.

**Figure 23**: Comparison of fluctuating pressure amplitudes at transducers S2 through S5, $\sigma = 2.5$.

**Figure 24**: Comparison of the total amplitudes of fluctuating pressures at $\sigma = 2.5$.

In Figure 23, the simulated fluctuating pressures exhibit a smoother distribution from transducer S2 to S5 than the experimental results. At all the five blade frequency harmonics and
on most transducers of impeller model P1 and P2, and at the 1\textsuperscript{st} blade frequency harmonics of impeller model P3, the fluctuating pressures are over predicted by numerical simulation; while at 2\textsuperscript{nd}–5\textsuperscript{th} blade frequency harmonics of impeller model P3, the fluctuating pressures are under predicted. So the total fluctuating pressure amplitudes at transducers S2–S5 are over predicted for impeller model P1 and P2 and under predicted for impeller model P3 by the CFD simulations, as shown in Figure 24.

The present numerical results also indicate that unloading towards the tip (P2 and P3) can effectively reduce the cavitation induced fluctuating pressures, which is the same as what was found in the experiments. Furthermore, the numerical simulations suggest that a tip rake towards upstream can reduce the fluctuating pressures on the tunnel wall.

4 CONCLUSIONS

Model tests have been carried out for the tunnel thruster in the cavitation tunnel of Shanghai Jiao Tong University to investigate the effects of the pitch and rake distributions of impeller blades on fluctuating pressures on the tunnel wall. Viscous flow CFD simulations are also performed at one cavitating condition. A number of conclusions are drawn according to the experimental and numerical results,

(1) In both non-cavitating and cavitating conditions, the fluctuating pressures are higher at the impeller disk than at upstream in general, but attenuate quickly as it goes downstream. The fluctuating pressures induced by impeller P1 are significantly higher than those by P2 and P3, indicating that the 'flat plate' blade is unfavorable for vibration excitation, and unloading towards the tip as often adopted in open propeller design should be introduced in the impeller design.

(2) CFD simulation is a useful tool for predicting the impeller vibration performance under cavitating condition and the cavity patterns are well predicted. However, the accuracy of predicted fluctuating pressures is yet to be improved.

(3) It is not yet clear to the authors how the rake distribution near the tip influences the tip clearance flow in the case of a tunnel thruster. The fluctuating pressures of impeller P2 are both lower than those of P3 based on the experiments, which is different from the results of our RANS simulations. Further study is needed in this respect.

REFERENCES


FLOW STUDY ON A DUCTED AZIMUTH THRUSTER

PATRICK SCHILLER*, KEQI WANG* AND MOUSTAFA ABDEL-MAKSOUD*

* Institute for Fluid Dynamics and Ship Theory (M-8), Hamburg University of Technology,
  Am Schwarzenberg-Campus 4, 21073 Hamburg, Germany
  e-mail: patrick.schiller@tu-harburg.de, url: http://www.tuhh.de/fds

Key words: Computational Methods, Marine Engineering, Ducted Azimuth Thruster

Abstract. This paper presents the results of the numerical validation and verification studies on an azimuth thruster. The numerical investigations include a grid study as well as an analysis of the simulation results obtained by different isotropic an anisotropic turbulence models, such as k-omega, SST, SAS-SST, BSL-EARSM and DES. The numerical simulation results of selected flow conditions are compared with experimental data. To investigate scale effects on the open water results numerical computations are carried out for a thruster in full- and model scale and the calculated thrust and torque coefficients are compared with model scale simulations and measurements.

1 INTRODUCTION

Azimuth Thrusters may experience considerable high dynamic loads due to operation in extreme off-design conditions. The flow on Azimuth Thrusters is highly unsteady due to large cavitation and separation areas, which may take place at strong oblique flow conditions. These strong oblique flows can be caused by high steering angles of the azimuth thruster, due to high drift angle of the ship or strong ocean currents. This may lead to high continuous changes in the amplitudes of the forces and moments and to flow separation on propeller blades and other components of the propulsion system, such as the shaft, the connecting struts between the nozzle and propeller hub or to the gondola.

The hydrodynamic behaviour of an azimuth thruster is studied within a Norwegian and German research project called Inter-Thrust. The project is carried out within the framework of MARTEC-II network under the lead of MARINTEK and participated by Havyard Ship Technology, Voith Turbo GmbH, Jastram GmbH and Hamburg University of Technology. In a first stage in the project different CFD modelling approaches with respect to grid resolution and turbulence modelling are investigated and subjected to validation and verification studies on a number of selected cases.

Advanced numerical methods based on the Navier-Stokes equations are particularly efficient in the simulation of the complex unsteady flow on ship propulsion systems such as nozzle propellers or POD drives [4][5]. In combination with multi-scale turbulence, e.g. Hybrid RANS-LES methods, CFD methods are capable of resolving strongly unsteady vortical structures in separated flow areas [6].

The validation and verification work has the purpose to develop best practice approaches regarding computation mesh, modeling of turbulence, and feasibility of engineering calculations over a large matrix of cases. Therefore, the validation and verification studies are
started with four simplified cases, for which detailed experimental data exist.

2 OBJECT OF INVESTIGATION

The investigated azimuth thruster is designed by MARINTEK. It was used at MARINTEK for experimental studies under various operation conditions. Thus, a lot of validation data is available. Figure 1 shows the thruster configuration.

![Model thruster configuration, experimental (left) and CAD (middle and right).](image)

The thruster housing, shaft and gondola, have a generic geometry and are manufactured in PVC. The model duct D-136 is used during the tests. This duct is of 19A type without diffuser. The duct is made of Plexiglas and has a length of 125 mm and is centered at propeller plane. The main specification of the duct is given in Table 1. The P-1374 model propeller is used in this thruster configuration and has also generic geometry designed by MARINTEK. The four-blade right-handed propeller can be used as ducted and open propeller and is made of aluminium alloy. Table 1 contains the main specification data of the propeller.

3 NUMERICAL METHODE

For this work, all numerical simulations are performed with the commercial CFD code ANSYS CFX (Release Version 15.0). It is a RANS method for steady and unsteady cases which numerically solves the unsteady equations of mass and momentum conservation:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0
\]

\[
\frac{\partial (\rho U)}{\partial t} + \nabla \cdot (\rho U \otimes U) = -\nabla p + \nabla \cdot \tau + S_M
\]

Where the stress tensor, \( \tau \), is related to the strain rate by:

\[
\tau = \mu \left( \nabla U + (\nabla U)^T - \frac{2}{3} \rho \nabla \cdot U \right)
\]

The ANSYS CFX solver uses second order discretization by default and is optimized for...
high performance computing. Different turbulence closure models (e.g. SST, SAS, DES and LES) are available. Free surfaces are handled via VOF based formulation and cavitation models based on Euler-Euler formulations are also available. For more details see [1].

4 SIMULATION CONDITIONS

4.1 Simulation Domain and Numerical Grid

All numerical grids are generated with ANSYS ICEM CFD meshing software. The computational domain consists of three parts:

- rotating propeller domain,
- thruster domain which includes the whole thruster geometry. The domain has a cylindrical shape and can be rotated around a vertical rotation axis of the thruster,
- exterior area.

The thruster domain has a diameter of 4D and a height of 3D. The exterior area has a quadratic form of 14D x 14D and a height of 7D. Figure 1 shows the used domain arrangement. For the consideration of different inflow angles towards the thruster the exterior area can be rotated. The free surface is neglected and the top of the domain is treated as free slip wall. Except the in- and the outlet, the other boundaries are treated as openings. The boundaries of the three domains are connected by sliding interfaces.

![Figure 2: Computational domain arrangement: top view, side view and perspective view.](image)

<table>
<thead>
<tr>
<th>Table 2: Cell numbers for the different grids.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid No. / Domain part</td>
</tr>
<tr>
<td>-------------------------</td>
</tr>
<tr>
<td>Grid 1 (2.7M)</td>
</tr>
<tr>
<td>Grid 2 (6M)</td>
</tr>
<tr>
<td>Grid 3 (20M)</td>
</tr>
<tr>
<td>Grid 4 (53M)</td>
</tr>
<tr>
<td>Grid 5 (27M)</td>
</tr>
</tbody>
</table>

Various structured grids with different characteristics have been generated, see Table 2. Grid number 1 to 4 are used for a grid study, whereby grid number 5 is generated for DES simulations, because they need a finer and more adapted grid regarding cell size and aspect ratio. The four meshes for the grid study follow a refinement factor of $1/\sqrt{2}$ in each spatial direction of the propeller and thruster domain. The cell number of the exterior area is kept
constant. Details of the grid arrangement can be seen in Figure 3. The $Y^+$ value is $\approx 1$ for grid 3.

![Figure 3: Details of the numerical grid, exemplary for grid 2.](image)

### 4.2 Flow Conditions

Four different operating conditions of the ducted thruster are considered for the grid study and the investigation of the performance of the different turbulence models. The conditions correspond to experiments performed at MARINTEK. The cases are characterized by the advance ration $J$, the rotational number of the propeller $n$ and the heading angle $\beta$ of the thruster, as listed in Table 3.

<table>
<thead>
<tr>
<th>Case No.</th>
<th>$J$ [-]</th>
<th>$n$ [rps, Hz]</th>
<th>Head. Angle $\beta$ [deg]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0</td>
<td>9</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0.6</td>
<td>11</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>0.6</td>
<td>9</td>
<td>-35</td>
</tr>
<tr>
<td>4</td>
<td>0.6</td>
<td>9</td>
<td>+35</td>
</tr>
</tbody>
</table>

For the validation of the of the open water characteristics additionally the following J-values at $n=11$Hz are simulated: $J=0.15; 0.3; 0.45; 0.75; 0.85; 0.95$ and 1.05. For the full-scale version, the same J-value are considered. The scaling factor is 20 and gives a propeller diameter of $D=5$m. The rotational speed of the propeller is $n=2.46$Hz for full-scale.

### 4.3 Turbulence Models

For all four operating conditions simulation are performed with different turbulence closure models. Following turbulence models have been used from ANSYS CFX:

- k-omega
- SST (as default)
- SAS-SST
- BSL-EARSM
- DES

The first two models (k-omega and SST) are isotropic models where the eddy viscosity is the same in each spatial direction. Whereas the other used models are anisotropic. For further details regarding the turbulence models see [1].

For the simulation with the DES model grid 5 has been generated, as mentioned before. This is due to the higher requirements regarding the cell sizes and aspect ratios for this turbulence model. For all other turbulence models grid 3 (20M cells) is used. The result from the SST model is used as initialization for the simulations with the SAS-SST, the BSL-EARSM and the
DES turbulence model.

4.4 Simulation Setup

In all simulations the timestep is set to 4° propeller rotation, except the simulations with the SAS-SST and DES turbulence model, here a timestep for 1° propeller rotation is used. For each timestep four inner iterations are considered. Several propeller revolutions are simulated till a periodic solution (quasi-steady) behaviour is achieved.

5 EVALUATION

For comparison with the experimental results obtained by MARINTEK different force and moment coefficients are calculated in a thruster fixed coordinate system for the propeller, the duct and the whole propulsion unit. The following values are evaluated:

- $k_t$ Propeller (KTP) Propeller Thrust
- $10k_q$ Propeller (10KQP) Propeller Torque
- $k_t$ Duct (KTD) Thrust of Duct
- $k_t$ Total (KTTOT) Total Thrust of the Unit
- $k_{side}$ Duct (KDS) Side Force of Duct
- $k_{side}$ Total (KSTOT) Total Side Force of the Unit
- $10k_Mz$ (10KMZ) Steering Moment (only shaft, gondola and propeller)

From the quasi steady solution result of each simulation an average over the last 1, 2 and 4 propeller revolution is calculated and normalized. The forces and moments are normalized as usual:

Forces by $\rho \times n^2 \times D^4$  \hspace{1cm} (4)

Moments by $\rho \times n^2 \times D^5$  \hspace{1cm} (5)

Additionally, a verification and validation (V&V) analysis [3] for the above mentioned values is carried out. The error ($E$) between CFD (S) and EFD (D) is expressed by following formula:

$$|E(\%D)| = \left| \frac{S - D}{D} \times 100 \right|$$  \hspace{1cm} (6)

6 SIMULATION RESULTS

Before the simulations results are analysed in detail some aspects should be mentioned. The first point is that for measuring the duct thrust in the model tests a separate shaft is used to connect the duct to the force balance. This shaft is not included in the thruster geometry considered in the numerical simulations, see Figure 1. This affects the mainly the total forces from the duct especially those at $\beta = \pm 35^\circ$.

A second point is that in the measurements the magnitude of the steering moment (10KMZ) at $\beta = \pm 35^\circ$ is quite differently, but one would expect values in the same order of magnitude. The simulation results as well as in other experimental investigations with the same thruster configuration without duct [2] (Fig 8) indicate that the steering moment is nearly symmetric for
both sides. So, there might be a problem in the measurements concerning the value for at β = +35°. Also, the values of the steering moment are quite small due to the well-balanced shaft, gondola and propeller arrangement.

6.1 Open water characteristics

As mentioned above, simulations are performed to determine the open water characteristic of the selected azimuth thruster unit and to compare them with the experimental results. Grid 2 (6M) is used and the thruster is simulated at above mentioned 7 J-values in model (MS) and in a full-scale (FS) version. The simulation results can be found in Figure 4 a-f.

The propeller thrust as well as the propeller torque (Figure 4 a and b) of the model scale variant agree very well with the experimental values. The KTP and 10KQP values for the full-scale are slightly smaller at low J-values and higher at huge J-values.

Regarding the duct thrust (Figure 4 c) the model scale values agree well in the range of J = 0.15 to 0.75 with the experimental. For J = 0 the value is slightly below the experiments whereas for J-values greater than 0.75 the values are clearly above. The values for the full-scale version separate from the experimental ones from J = 0.6 and the distance to greater values grows. The differences between the three curves at high J-values come from different flow separation areas at the leading edge of the duct as indicated in Figure 7.

Figure 4 d shows the comparison of the housing resistance (shaft + gondola). Here the three curves are quite differently but the general trend shows a lower resistance with increasing J-values. The full-scale variant has the lowest drag. The experimental curve has the same tendency as the full-scale version but shifted to higher drag values. The calculated values for model scale are till J = 0.45 lower than the experimental ones. From J = 0.45 to 0.95 they are almost identical and after that they are higher. These differences might come from differently predicted flow separation area due to not sufficient mesh resolution in this region.

The comparison of the total thrust produced by the whole thruster unit can be found in Figure 4 e. The value for the three curves are till J = 0.6 almost identical. Beyond that the full-scale variant shows higher value due to the higher duct thrust. This can also be seen in the total
thruster efficiency illustrated in Figure 4 f. Here the full-scale version reaches the highest efficiency over a greater range.

Further the scale effects between the calculated model scale and the full-scale thruster version are analysed. The differences ($\Delta = (FS - MS)/MS \times 100\%$) are presented in Figure 5 a-e for the different coefficients. The scale effects on the propeller thrust und torque (Figure 5 a and b) show in general the same trend. The coefficients are lower for full scale which leads to a negative difference. These increases slightly till $J = 0.15$ and then continuously decreases. The differences are greater for the propeller torque so that this results in a better efficiency for the full-scale version in total. A comparison of the pressure distribution and limiting streamlines on the propeller at $J = 0$ and $J = 0.6$ can be found in Figure 6 left. Clearly visible is that the flow in the model scale case cannot provide enough shear force against centrifugal force; hence the streamlines on both blade sides are not travelling along the circumferential direction but through the various radii.

Figure 5 c illustrates the scale effect regarding the duct thrust. The difference is positive indicating a greater thrust for FS and lies around about 5% till $J = 0.45$. Beyond that the delta increases rapidly to value over 20% because the produced thrust of the duct in MS at high $J$-values becomes small and even negative. As mentioned before the differences at high $J$-values come from different flow separation regions at the leading edge of the duct as indicated in Figure 7.

![Figure 5: Scale effects for different advance ratios on a) propeller thrust (KTP); b) propeller torque (KQP) c) duct thrust (KTD); d) housing resistance (KTG) and e) total thrust (KTTOT).](image)

The scale effect regarding the housing resistance, depicted in Figure 5 d, is continuously increasing from about 7% at $J = 0$ to 40% at $J = 0.75$. The difference is negative which means that the resistance is lower for the full-scale version. This increasing scale effect is caused mainly by separation effects on the shaft. Figure 6 right show the pressure distribution and limiting streamlines on the shaft and the gondola where this fact is explicit visible. The separation zone is in FS smaller than in MS which leads to a smaller resistance. The flow separation is induced by the high positive pressure gradient in this region. Due to the higher suction impact of the propeller at low $J$-values the separation zones are smaller at low inflow velocities.

With respect to the total thrust produced be the azimuth thruster (Figure 5 e) one can see
after a slight reduction at \( J = 0.15 \) a progressive increasing scale effect with increasing \( J \)-value. This mainly caused by the greater duct and propeller thrust in full-scale.

6.2 Validation

The validation and verification work has the purpose of development of best practice approaches regarding computation mesh resolution, modelling of turbulence, and feasibility of engineering calculations over a large matrix of cases. Therefore, the validation and verification studies are started with four simplified cases, for which detailed experimental data exist.

Grid resolution

All four above described cases are calculated on grid 1 to 4. The maximal occurred \( Y^+ \)-value for all investigated flow conditions during the simulation is 7, 5, 3.5 and 2.5 for grid 1, 2, 3 and 4 respectively. The calculated coefficient values for case 1 to 4 can be found in the Table 4 to Table 7. Also, the error of the simulation values with respect to the experimental ones are shown in the tables.

For case 1 and 2 the propeller thrust and torque coefficients agrees very well for all grid resolutions with respect to the experimental ones. They lie in-between a deviation of maximal 2%. The thrust coefficient of the duct is in case 1 underpredicted by 4-8%, whereby the difference decreases with higher mesh resolution. Whereas the duct thrust is overpredicted
about 12-16% in case 2. The deviation is for the grids 2, 3 and 4 nearly the same. A possible reason for this high deviation might be the missing resistance from the mounting of the duct to the force balance in the computation. Because of the variation in the duct thrust the same tendencies can be found in the total thrust of unit. The deviation is 1-4% below the measured values for case 1 and 2-6% above for case 2.

Due to the heading angle of $\beta = 0^\circ$ for case 1 and 2 the duct and total unit side force coefficients are very small. The small values of the forces can lead to large deviations which is here the case. Therefore, the values are not further assessed here.

Table 4: Simulation results of grid resolution study for case 1, $J=0.0$, $n=9$Hz, $\beta =0^\circ$.

<table>
<thead>
<tr>
<th>Setup</th>
<th>$kt$ Propeller</th>
<th>$10kg$ Propeller</th>
<th>kt Duct</th>
<th>kt Total</th>
<th>Side Duct</th>
<th>Side Total</th>
<th>ISM Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.7M SST</td>
<td>0.3300</td>
<td>0.6000</td>
<td>0.3360</td>
<td>0.6340</td>
<td>0.0030</td>
<td>-0.1530</td>
<td>0.0150</td>
</tr>
<tr>
<td>6M SST</td>
<td>0.3260</td>
<td>0.5935</td>
<td>0.3268</td>
<td>0.6101</td>
<td>0.0015</td>
<td>0.0035</td>
<td>0.0182</td>
</tr>
<tr>
<td>20M SST</td>
<td>0.3287</td>
<td>0.5936</td>
<td>0.3350</td>
<td>0.6214</td>
<td>0.0006</td>
<td>0.0030</td>
<td>0.0204</td>
</tr>
<tr>
<td>53M SST</td>
<td>0.3010</td>
<td>0.5998</td>
<td>0.3427</td>
<td>0.6273</td>
<td>0.0008</td>
<td>0.0033</td>
<td>0.0176</td>
</tr>
<tr>
<td>2.7M SST</td>
<td>-0.4</td>
<td>-0.8</td>
<td>-8.2</td>
<td>-3.8</td>
<td>-50.0</td>
<td>-126.4</td>
<td>21.6</td>
</tr>
<tr>
<td>6M SST</td>
<td>-0.7</td>
<td>-1.1</td>
<td>-4.6</td>
<td>-2.0</td>
<td>-72.7</td>
<td>-121.2</td>
<td>36.1</td>
</tr>
<tr>
<td>20M SST</td>
<td>-0.2</td>
<td>0.8</td>
<td>-3.7</td>
<td>-1.1</td>
<td>-72.3</td>
<td>-123.8</td>
<td>17.3</td>
</tr>
<tr>
<td>53M SST</td>
<td>0.1</td>
<td>-1.0</td>
<td>12.2</td>
<td>2.5</td>
<td>-134.7</td>
<td>-42.9</td>
<td></td>
</tr>
<tr>
<td>2.7M SST</td>
<td>0.1</td>
<td>0.1</td>
<td>16.5</td>
<td>4.3</td>
<td>-131.9</td>
<td>-30.5</td>
<td></td>
</tr>
<tr>
<td>6M SST</td>
<td>0.6</td>
<td>0.7</td>
<td>15.9</td>
<td>5.2</td>
<td>-130.9</td>
<td>-20.3</td>
<td></td>
</tr>
<tr>
<td>20M SST</td>
<td>0.8</td>
<td>1.0</td>
<td>16.3</td>
<td>6.0</td>
<td>-124.5</td>
<td>-6.8</td>
<td></td>
</tr>
<tr>
<td>53M SST</td>
<td>0.8</td>
<td>1.0</td>
<td>16.3</td>
<td>6.0</td>
<td>-124.5</td>
<td>-6.8</td>
<td></td>
</tr>
</tbody>
</table>

Table 5: Simulation results of grid resolution study for case 2, $J=0.6$, $n=11$Hz, $\beta =0^\circ$.

<table>
<thead>
<tr>
<th>Setup</th>
<th>$kt$ Propeller</th>
<th>$10kg$ Propeller</th>
<th>kt Duct</th>
<th>kt Total</th>
<th>Side Duct</th>
<th>Side Total</th>
<th>ISM Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.7M SST</td>
<td>0.2020</td>
<td>0.4060</td>
<td>0.2180</td>
<td>0.4320</td>
<td>0.0100</td>
<td>0.2820</td>
<td>0.0410</td>
</tr>
<tr>
<td>6M SST</td>
<td>0.1973</td>
<td>0.4072</td>
<td>0.2122</td>
<td>0.4308</td>
<td>0.0104</td>
<td>0.2804</td>
<td>0.0419</td>
</tr>
<tr>
<td>20M SST</td>
<td>0.2144</td>
<td>0.4175</td>
<td>0.3168</td>
<td>0.5365</td>
<td>0.0167</td>
<td>0.3567</td>
<td>0.0662</td>
</tr>
<tr>
<td>53M SST</td>
<td>0.2124</td>
<td>0.4191</td>
<td>0.3175</td>
<td>0.5366</td>
<td>0.0165</td>
<td>0.3563</td>
<td>0.0667</td>
</tr>
<tr>
<td>2.7M SST</td>
<td>-2.5</td>
<td>-3.2</td>
<td>-5.7</td>
<td>-3.9</td>
<td>-10.2</td>
<td>-2.9</td>
<td>-8.4</td>
</tr>
<tr>
<td>6M SST</td>
<td>1.5</td>
<td>2.8</td>
<td>6.9</td>
<td>2.5</td>
<td>-10.2</td>
<td>-13.2</td>
<td>21.5</td>
</tr>
<tr>
<td>20M SST</td>
<td>5.2</td>
<td>3.2</td>
<td>7.4</td>
<td>0.9</td>
<td>-9.5</td>
<td>-9.9</td>
<td>9.1</td>
</tr>
<tr>
<td>53M SST</td>
<td>5.2</td>
<td>3.2</td>
<td>7.4</td>
<td>0.9</td>
<td>-9.5</td>
<td>-9.9</td>
<td>9.1</td>
</tr>
</tbody>
</table>

Table 6: Simulation results of grid resolution study for case 3, $J=0.6$, $n=9$Hz, $\beta =-35^\circ$.

<table>
<thead>
<tr>
<th>Setup</th>
<th>$kt$ Propeller</th>
<th>$10kg$ Propeller</th>
<th>kt Duct</th>
<th>kt Total</th>
<th>Side Duct</th>
<th>Side Total</th>
<th>ISM Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.7M SST</td>
<td>0.3590</td>
<td>0.5435</td>
<td>0.1940</td>
<td>0.4280</td>
<td>-0.0300</td>
<td>-0.0170</td>
<td>0.0000</td>
</tr>
<tr>
<td>6M SST</td>
<td>0.3460</td>
<td>0.5460</td>
<td>0.1995</td>
<td>0.4215</td>
<td>-0.0250</td>
<td>-0.0180</td>
<td>0.0000</td>
</tr>
<tr>
<td>20M SST</td>
<td>0.3462</td>
<td>0.5461</td>
<td>0.1995</td>
<td>0.4215</td>
<td>-0.0250</td>
<td>-0.0180</td>
<td>0.0000</td>
</tr>
<tr>
<td>53M SST</td>
<td>0.3362</td>
<td>0.5461</td>
<td>0.1995</td>
<td>0.4215</td>
<td>-0.0250</td>
<td>-0.0180</td>
<td>0.0000</td>
</tr>
<tr>
<td>2.7M SST</td>
<td>-3.1</td>
<td>-4.1</td>
<td>-5.3</td>
<td>-4.8</td>
<td>-16.0</td>
<td>-15.0</td>
<td>-189.7</td>
</tr>
<tr>
<td>6M SST</td>
<td>-2.1</td>
<td>1.3</td>
<td>-2.4</td>
<td>-0.6</td>
<td>-18.1</td>
<td>-24.2</td>
<td>555.7</td>
</tr>
<tr>
<td>20M SST</td>
<td>-1.7</td>
<td>3.0</td>
<td>-7.7</td>
<td>-1.4</td>
<td>-11.2</td>
<td>-18.7</td>
<td>761.0</td>
</tr>
<tr>
<td>53M SST</td>
<td>-0.3</td>
<td>3.3</td>
<td>-3.3</td>
<td>0.7</td>
<td>-11.7</td>
<td>-18.4</td>
<td>758.7</td>
</tr>
</tbody>
</table>

Table 7: Simulation results of grid resolution study for case 4, $J=0.6$, $n=9$Hz, $\beta =+35^\circ$.

<table>
<thead>
<tr>
<th>Setup</th>
<th>$kt$ Propeller</th>
<th>$10kg$ Propeller</th>
<th>kt Duct</th>
<th>kt Total</th>
<th>Side Duct</th>
<th>Side Total</th>
<th>ISM Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.7M SST</td>
<td>0.2907</td>
<td>0.5645</td>
<td>0.1535</td>
<td>0.4638</td>
<td>-0.2562</td>
<td>-0.3801</td>
<td>-0.0072</td>
</tr>
<tr>
<td>6M SST</td>
<td>0.2937</td>
<td>0.5489</td>
<td>0.1601</td>
<td>0.4256</td>
<td>-0.2497</td>
<td>-0.3373</td>
<td>0.0525</td>
</tr>
<tr>
<td>20M SST</td>
<td>0.2949</td>
<td>0.5329</td>
<td>0.1648</td>
<td>0.4295</td>
<td>-0.2708</td>
<td>-0.3636</td>
<td>0.0689</td>
</tr>
<tr>
<td>53M SST</td>
<td>0.2992</td>
<td>0.5461</td>
<td>0.1695</td>
<td>0.4312</td>
<td>-0.2650</td>
<td>-0.3646</td>
<td>0.0687</td>
</tr>
<tr>
<td>2.7M SST</td>
<td>-3.1</td>
<td>-4.1</td>
<td>-5.3</td>
<td>-4.8</td>
<td>-16.0</td>
<td>-15.0</td>
<td>-189.7</td>
</tr>
<tr>
<td>6M SST</td>
<td>-2.1</td>
<td>1.3</td>
<td>-2.4</td>
<td>-0.6</td>
<td>-18.1</td>
<td>-24.2</td>
<td>555.7</td>
</tr>
<tr>
<td>20M SST</td>
<td>-1.7</td>
<td>3.0</td>
<td>-7.7</td>
<td>-1.4</td>
<td>-11.2</td>
<td>-18.7</td>
<td>761.0</td>
</tr>
<tr>
<td>53M SST</td>
<td>-0.3</td>
<td>3.3</td>
<td>-3.3</td>
<td>0.7</td>
<td>-11.7</td>
<td>-18.4</td>
<td>758.7</td>
</tr>
</tbody>
</table>

Also, due to the well-balanced shaft, gondola and propeller arrangement small steering moment values are calculated, see the results of case 1 and 2. In both cases the quality of the steering moment results is improved significantly with increasing grid resolution.

The deviation of the coefficients for propeller thrust and torque, duct thrust as well as total thrust of the unit in general lie for the cases 3 and 4 in a range of max. ±5%. The largest
deviation can be found in the case of grid 1 with the lowest cell number. The other three grids show nearly the same deviations. For case 3 the predicted values are in general slightly larger, whereas the values for case 4 are slightly smaller, particularly the values of the propeller thrust.

The calculated values for the side forces (duct and total) are underpredicted. The deviations to the experiment is around 10-20%, whereas the deviations are a bit higher for case 4. Keeping in mind that the duct mounting connection to the force balance, which creates additional side force, is missing in the simulation, the prediction accuracy seems to be within acceptable range. No clear trend is recognizable regarding the mesh resolutions for grid 2, 3 and 4 because the deviations lie in the same range.

This is not the case at least for the deviation of the steering moment at case 3. Here the accuracy of the predicted results gets higher with finer mesh resolution. The smallest deviation is 9% at grid 4. For case 4 it is difficult to conclude a clear statement. As mentioned above the measured value might be defective which leads to very high and illogical deviations.

**Turbulence Models**

All four flow cases are calculated with the above mentioned turbulence models. Except for the DES model, where grid 5 is employed, grid 3 is used. The calculated coefficient values for case 1 to 4 can be found in the Table 8 to Table 11. Again, the errors between the simulation and the experiments are shown in the tables.

For case 1 and 2 the propeller thrust is well predicted by all applied turbulence models within a deviation of 1% with the exception of the k-omega and DES model where the deviation is up to 3%. Also, the propeller torque is very well predicted for case 1 and 2 by all turbulence models without any exception. The deviation lies here in a range of 1%.

All turbulence models underpredict the duct thrust for case 1 by about 5% and overpredict it for case 2 by about 14%. This deviation is quite high, but acceptable due to the missing consideration of the mounting resistance of the duct in the computation. Because the total thrust of the unit is composed mainly of the propeller and duct thrust the same tendencies can be found here. The deviation is 2-4% below the measured values for case 1 and 2-5% above for case 2.

Due to the fact that for case 1 and 2 the values of the duct and total unit side force as well as for the steering moment are very small, the deviations are quite large. Also, no clear trend can be found regarding the turbulence modelling. Therefore, the values are not further assessed here.

For the cases with \(\beta=\pm35^\circ\) heading angle the propeller thrust is overpredicted about 3-6% for case 3 and underpredicted about 2-6% for case 4. Again, the k-omega and DES model show slightly smaller thrust coefficients than the other models. The propeller torque is predicted by all turbulence models quite well for both cases (3 and 4). The deviation is 2-5% above the measured ones.

The deviation for the duct thrust is 3-7% above the experimental values for case 3 and around 2-3% for case 4. Also, here the k-omega and DES model show slightly smaller duct thrust coefficients than the other models. Due to the marginally over predicted propeller and duct thrust for case 3 also the total thrust is overpredicted by 3-8%. For case 4 the prediction quality is better and the deviation is below 2% because of the underpredicted propeller thrust.

The calculated values for the duct and total side forces are comparably underpredicted by all turbulence models. The deviations to the experiment is around 10-12% for the duct side force and 14-20% for the total side force. For case 4 the deviations are a bit higher than for case 3.
As mentioned above due to the missing duct mounting in the simulation, a validation of the turbulence modelling. Here the k-omega and DES model significantly under predict the steering moment. The deviation is at least 25%. The other models overpredict the by 14-21%. For case 4 it is difficult to make a clear statement, as the measured value might be defective.

Table 8: Simulation results of turbulence model variation for case 1, J=0.0, n=9Hz, β =0°.

<table>
<thead>
<tr>
<th>Setup</th>
<th>k Propeller</th>
<th>10k Propeller</th>
<th>duct Duct</th>
<th>Total Duct</th>
<th>10Duct</th>
<th>10M Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>0.3300</td>
<td>0.8690</td>
<td>0.3540</td>
<td>0.0140</td>
<td>0.0005</td>
<td>-0.0170</td>
</tr>
<tr>
<td>20 M SST</td>
<td>0.3278</td>
<td>0.5936</td>
<td>0.3395</td>
<td>0.6214</td>
<td>0.0081</td>
<td>-0.0010</td>
</tr>
<tr>
<td>20 M k-omega</td>
<td>0.3200</td>
<td>0.5996</td>
<td>0.3319</td>
<td>0.6089</td>
<td>0.0025</td>
<td>-0.0053</td>
</tr>
<tr>
<td>20 M BSL-EARSM</td>
<td>0.3283</td>
<td>0.5948</td>
<td>0.3388</td>
<td>0.6218</td>
<td>0.0034</td>
<td>0.0045</td>
</tr>
<tr>
<td>20 M SAS-SST</td>
<td>0.3220</td>
<td>0.5946</td>
<td>0.3389</td>
<td>0.6233</td>
<td>0.0021</td>
<td>0.0016</td>
</tr>
<tr>
<td>27 M DES</td>
<td>0.3197</td>
<td>0.5940</td>
<td>0.3371</td>
<td>0.6136</td>
<td>0.0018</td>
<td>0.0142</td>
</tr>
<tr>
<td>20 M SST</td>
<td>-0.7</td>
<td>-1.1</td>
<td>-4.6</td>
<td>-2.3</td>
<td>-7.7</td>
<td>-122.2</td>
</tr>
<tr>
<td>20 M k-omega</td>
<td>-1.3</td>
<td>-0.3</td>
<td>-1.3</td>
<td>-4.0</td>
<td>-14.1</td>
<td>-49.7</td>
</tr>
<tr>
<td>20 M BSL-EARSM</td>
<td>-0.5</td>
<td>-0.9</td>
<td>-4.8</td>
<td>-1.9</td>
<td>12.9</td>
<td>25.6</td>
</tr>
<tr>
<td>20 M SAS-SST</td>
<td>-0.5</td>
<td>-0.9</td>
<td>-4.8</td>
<td>-2.0</td>
<td>30.2</td>
<td>253.5</td>
</tr>
<tr>
<td>27 M DES</td>
<td>-3.1</td>
<td>-1.0</td>
<td>-5.3</td>
<td>-3.2</td>
<td>266.0</td>
<td>299.1</td>
</tr>
</tbody>
</table>

Table 9: Simulation results of turbulence model variation for case 2, J=0.6, n=11Hz, β =0°.

<table>
<thead>
<tr>
<th>Setup</th>
<th>k Propeller</th>
<th>10k Propeller</th>
<th>duct Duct</th>
<th>Total Duct</th>
<th>10Duct</th>
<th>10M Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>0.2770</td>
<td>0.4900</td>
<td>0.5960</td>
<td>0.2740</td>
<td>0.0000</td>
<td>0.0200</td>
</tr>
<tr>
<td>20 M SST</td>
<td>0.2585</td>
<td>0.4946</td>
<td>0.5969</td>
<td>0.2904</td>
<td>0.0029</td>
<td>0.0040</td>
</tr>
<tr>
<td>20 M k-omega</td>
<td>0.2497</td>
<td>0.4991</td>
<td>0.5965</td>
<td>0.2905</td>
<td>0.0027</td>
<td>0.0033</td>
</tr>
<tr>
<td>20 M BSL-EARSM</td>
<td>0.2549</td>
<td>0.4992</td>
<td>0.5968</td>
<td>0.2907</td>
<td>0.0021</td>
<td>0.0034</td>
</tr>
<tr>
<td>20 M SAS-SST</td>
<td>0.2509</td>
<td>0.4989</td>
<td>0.5964</td>
<td>0.2879</td>
<td>0.0048</td>
<td>0.0073</td>
</tr>
<tr>
<td>27 M DES</td>
<td>0.2508</td>
<td>0.4953</td>
<td>0.5965</td>
<td>0.2817</td>
<td>0.0027</td>
<td>0.0026</td>
</tr>
<tr>
<td>20 M SST</td>
<td>0.6</td>
<td>0.7</td>
<td>13.9</td>
<td>5.2</td>
<td>-130.6</td>
<td>-26.3</td>
</tr>
<tr>
<td>20 M k-omega</td>
<td>-2.2</td>
<td>1.2</td>
<td>8.8</td>
<td>2.4</td>
<td>-125.8</td>
<td>-34.2</td>
</tr>
<tr>
<td>20 M BSL-EARSM</td>
<td>0.8</td>
<td>1.3</td>
<td>13.7</td>
<td>5.3</td>
<td>-126.2</td>
<td>-47.8</td>
</tr>
<tr>
<td>20 M SAS-SST</td>
<td>1.1</td>
<td>1.0</td>
<td>13.2</td>
<td>4.3</td>
<td>-156.0</td>
<td>-47.8</td>
</tr>
<tr>
<td>27 M DES</td>
<td>-2.4</td>
<td>0.5</td>
<td>8.1</td>
<td>2.1</td>
<td>-119.7</td>
<td>-32.8</td>
</tr>
</tbody>
</table>

Table 10: Simulation results of turbulence model variation for case 3, J=0.6, n=9Hz, β =-35°.

<table>
<thead>
<tr>
<th>Setup</th>
<th>k Propeller</th>
<th>10k Propeller</th>
<th>duct Duct</th>
<th>Total Duct</th>
<th>10Duct</th>
<th>10M Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>0.2120</td>
<td>0.4080</td>
<td>0.3120</td>
<td>0.2482</td>
<td>0.0101</td>
<td>0.0572</td>
</tr>
<tr>
<td>20 M SST</td>
<td>0.2114</td>
<td>0.4173</td>
<td>0.3136</td>
<td>0.2855</td>
<td>0.0233</td>
<td>0.1567</td>
</tr>
<tr>
<td>20 M k-omega</td>
<td>0.2085</td>
<td>0.4284</td>
<td>0.3129</td>
<td>0.2558</td>
<td>0.0237</td>
<td>0.1639</td>
</tr>
<tr>
<td>20 M BSL-EARSM</td>
<td>0.2136</td>
<td>0.4318</td>
<td>0.3135</td>
<td>0.2507</td>
<td>0.0207</td>
<td>0.1556</td>
</tr>
<tr>
<td>20 M SAS-SST</td>
<td>0.2133</td>
<td>0.4207</td>
<td>0.3176</td>
<td>0.3053</td>
<td>0.0253</td>
<td>0.1541</td>
</tr>
<tr>
<td>27 M DES</td>
<td>0.2105</td>
<td>0.4245</td>
<td>0.3127</td>
<td>0.3142</td>
<td>0.0233</td>
<td>0.1541</td>
</tr>
<tr>
<td>20 M SST</td>
<td>4.9</td>
<td>2.8</td>
<td>6.9</td>
<td>2.5</td>
<td>-17.2</td>
<td>-32.1</td>
</tr>
<tr>
<td>20 M k-omega</td>
<td>3.2</td>
<td>3.5</td>
<td>3.9</td>
<td>8.2</td>
<td>-10.0</td>
<td>-52.3</td>
</tr>
<tr>
<td>20 M BSL-EARSM</td>
<td>5.7</td>
<td>4.2</td>
<td>8.1</td>
<td>2.9</td>
<td>-12.8</td>
<td>-52.3</td>
</tr>
<tr>
<td>20 M SAS-SST</td>
<td>6.6</td>
<td>3.6</td>
<td>7.5</td>
<td>3.1</td>
<td>-10.3</td>
<td>-32.1</td>
</tr>
<tr>
<td>27 M DES</td>
<td>4.2</td>
<td>4.6</td>
<td>3.6</td>
<td>4.4</td>
<td>-10.2</td>
<td>-24.5</td>
</tr>
</tbody>
</table>

Table 11: Simulation results of turbulence model variation for case 4, J=0.6, n=9Hz, β =+35°.

<table>
<thead>
<tr>
<th>Setup</th>
<th>k Propeller</th>
<th>10k Propeller</th>
<th>duct Duct</th>
<th>Total Duct</th>
<th>10Duct</th>
<th>10M Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>0.2949</td>
<td>0.3529</td>
<td>0.1640</td>
<td>0.4281</td>
<td>-0.0501</td>
<td>-0.4430</td>
</tr>
<tr>
<td>20 M SST</td>
<td>0.2816</td>
<td>0.3552</td>
<td>0.1606</td>
<td>0.4318</td>
<td>-0.0533</td>
<td>-0.3632</td>
</tr>
<tr>
<td>20 M k-omega</td>
<td>0.2954</td>
<td>0.3550</td>
<td>0.1644</td>
<td>0.4242</td>
<td>-0.0468</td>
<td>-0.3520</td>
</tr>
<tr>
<td>20 M BSL-EARSM</td>
<td>0.2962</td>
<td>0.3530</td>
<td>0.1686</td>
<td>0.4193</td>
<td>-0.2704</td>
<td>-0.3586</td>
</tr>
<tr>
<td>20 M SAS-SST</td>
<td>0.2835</td>
<td>0.3474</td>
<td>0.1595</td>
<td>0.4232</td>
<td>-0.2683</td>
<td>-0.3668</td>
</tr>
<tr>
<td>27 M DES</td>
<td>-1.7</td>
<td>-2.0</td>
<td>-2.7</td>
<td>-0.4</td>
<td>-11.3</td>
<td>-18.7</td>
</tr>
<tr>
<td>20 M SST</td>
<td>-6.1</td>
<td>2.1</td>
<td>-2.1</td>
<td>0.9</td>
<td>-13.0</td>
<td>-47.3</td>
</tr>
<tr>
<td>20 M k-omega</td>
<td>-2.2</td>
<td>2.0</td>
<td>0.2</td>
<td>-0.9</td>
<td>-13.2</td>
<td>-21.3</td>
</tr>
<tr>
<td>20 M BSL-EARSM</td>
<td>-1.2</td>
<td>2.0</td>
<td>2.8</td>
<td>-2.0</td>
<td>-11.3</td>
<td>-21.3</td>
</tr>
<tr>
<td>27 M DES</td>
<td>-5.5</td>
<td>-1.0</td>
<td>-2.8</td>
<td>-1.1</td>
<td>-13.5</td>
<td>-64.4</td>
</tr>
</tbody>
</table>

7 CONCLUSIONS

In this work a generic azimuth thruster was investigated at different flow conditions. The
open water characteristics were calculated for model and full-scale configuration and compared with experimental values. Further a grid study with four different mesh resolutions was conducted for four diverse flow conditions. Finally, different isotropic and anisotropic turbulence models were compared for the four selected operating conditions.

The comparison of the calculated open water characteristics in model scale show in general a fair agreement with the measurement. Some deviations arise at high J-values probably due to not correctly captured separation phenomena on the duct and on the thruster shaft. Scale effects can be evaluated based on the calculated results of model and full-scale. Affected are mainly the duct thrust and the housing resistance due to separation phenomena particularly at high J-values. In total the scale effects lead to a higher efficiency over a larger advance ratio range for the full-scale version.

Regarding the results of the conducted grid study it can be said that the grids from and above 6M cells for the investigated operating conditions give good results. Also for “not complicated” flow conditions as for example for open water characteristic good results can be achieved on coarser grids, but partly and especially for the steering moment higher deviations can occur. Further it was ascertained that there is no significant accuracy increase with a finer grid resolution. The variation of the turbulence model has shown that no turbulence model is significantly better than another one for the here investigated azimuth thruster and flow conditions. All of them show a fair agreement with experimental results. This could be different if operating conditions are simulated with a huge amount of separated flow. The results of the k-omega and DES turbulence model reveal partly slightly lower coefficient values than the other ones.

In summary, the comparison of the simulation results with the model tests has shown that for the calculated cases of the investigated azimuth thruster a grid size of 6 to 10 million cells and the use of the SST turbulence model are sufficient to achieve reasonable accuracy at least for industrial applications. With finer grids and other turbulence models, partly slightly better results can be achieved, but they do not justify the considerably higher computational effort during a design stage.

8 REFERENCES

NUMERICAL ASSESSMENT OF PROPELLER-HULL INTERACTION AND PROPELLER HUB EFFECTS FOR A TWIN SCREW VESSEL

MARINE 2017

HEINRICH STRECKWALL*, YAN XING-KAEDING*, THOMAS LÜCKE*, TOMASZ BUGALSKI†, TOM GOEDICKE‡ AND ALAZ TALAY§

* HSVA, Bramfelder Str. 164, 22305 Hamburg/Germany
e-mail: streckwall@hsva.de

† CTO, Szczecińska 65, 80-392 Gdańsk/Poland
e-mail: tomasz.bugalski@cto.gda.pl

‡ MMG, Teterower Str. 1, 17192 Waren (Müritz)/Germany
e-mail: goedicke@mmg-propeller.de

§ MILPER, Sanayi Mah.Teknopark Blvd. 1, Pendik 34906, Istanbul/Turkey
e-mail: alaz.talay@milper.com.tr

Key words: Relative rotative efficiency, propeller hub, hub vortex, self-propulsion, twin screw.

Abstract. A numerical study that addresses twin screw propulsion was conducted and results using the RANS solvers ‘FreSCo+’ and ‘Fluent’ were shared. In order to avoid potential problems on property rights we combined the DTMB (David Taylor Model Basin) model No. 5415 and the SVA (Potsdam Model Basin) propeller No. CPP 1304. The computational self-propulsion point was identified via a numerical implementation of the so-called ‘British Method’. In this particular case, linked to the hub dimensions of the chosen propeller, the detailed modelling of the propeller hub and the true resolution of its connection to the hull was rather important. The same view holds for the propeller open water test setup. For the latter case we learned that the comparison with uncorrected experimental thrust data could represent a better way to confirm the numerical results.

1 INTRODUCTION

Experimental and numerical results on the twin screw vessel model DTMB 5415 usually address its manoeuvring performance (see e.g. [1]). Data on pure self-propulsion are rare for DTMB 5415 and the best one can get from literature on thrust and torque at the installed propeller may be found in reference [2]. However we decided to base a common numerical self-propulsion study on DTMB 5415 using the RANS solvers ‘FreSCo+’ [3] and ‘Fluent’. The integration of a suitable twin screw propeller, which not only had to fit to operational requests but also had to satisfy CAD needs on blade surface and hub, was one of our
demanding tasks. Viewing the quality of the CAD representation and the unrestricted availability of the CAD data, the SVA (Potsdam Model Basin) propeller CPP1304 was chosen for such a comparative study run in different institutions.

It was agreed to rely on Double Body (DB) setups as the modelling of the rotating propeller introduced already a sufficient amount of complexity. Due to the general lack of propulsion test data, self-propulsion shaft frequencies suitable for CPP 1304 behind DTMB model 5415 were rather to be evaluated than to be prescribed. However we maintained the DB approach and the final scheme to relate a given ship speed to the propeller shaft frequency resembles the ‘British Method’ known from towing tank tests. Accordingly in our simulation the propulsion point is not directly met but interpolated from DB runs with fixed shaft frequency, representing overload and underload settings. Introducing the ‘British Method’ to control a numerical process one may save time and resources.

2 PROPELLER OPEN WATER TEST AND RESULTS FROM CFD SIMULATION

The findings from this section suggest a certain caution when comparing RANS simulations on open water (OW) tests and related experiments. Similar to the RANS treatment of propulsion tests discussed further below we set emphasis on the propeller hub details entering the numerical grids. For the evaluation of the numerically treated OW-mode one should request ‘uncorrected’ thrust data from the OW tests. Such uncorrected data where available for CPP 1304 and finally entered the comparison of measured and calculated OW curves.

2.1 Propeller 1304 as an example for hub effects in Open Water mode

The larger the boss dimensions of the propeller the higher the risk to invoke errors when doing manipulations in the hub area for convenience. This caution was already stressed by the authors in a contribution to NuTTS’16 [4]. Geometrical as well as experimental data for the CPP 1304 of the Potsdam Model Basin (SVA) were available from the Potsdam website. This propeller served for several benchmarks on OW computation, cavitation analysis and (latest) propeller-scaling (see for instance [5]). The geometry was documented including all details on shafting, nose cap and on the gap between rotating and fixed parts of the OW-setup.

<table>
<thead>
<tr>
<th>Type</th>
<th>FS (behind)</th>
<th>FS (OW)</th>
<th>Model Scale (behind)</th>
<th>Model Scale (OW)</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of blades</td>
<td>CP</td>
<td>CP</td>
<td>CP</td>
<td>CP</td>
</tr>
<tr>
<td>D (m)</td>
<td>5</td>
<td>5</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>P/D (0.7R)</td>
<td>5.10</td>
<td>3</td>
<td>0.2055</td>
<td>0.25</td>
</tr>
<tr>
<td>Ae/A0</td>
<td>1.635</td>
<td>1.635</td>
<td>1.635</td>
<td>1.635</td>
</tr>
<tr>
<td>Rotation</td>
<td>0.779</td>
<td>0.779</td>
<td>0.779</td>
<td>0.779</td>
</tr>
<tr>
<td>Hub ratio</td>
<td>inw/outw</td>
<td>right</td>
<td>inw/outw</td>
<td>right</td>
</tr>
<tr>
<td></td>
<td>0.3</td>
<td>0.3</td>
<td>0.3</td>
<td>0.3</td>
</tr>
</tbody>
</table>

Table 1: Particulars propeller 1304 (*‘Model Scale OW’ existing as hardware; other scales are hypothetic*)
Table 1 gives the main particulars of CPP 1304. Only the column titled ‘Model Scale OW’ relates to an existing hardware. The other scales are hypothetic as the table also reflects dimensions related to full scale OW calculations and to a ‘fitted’ version of CPP 1304 (smaller diameter) which is entering in-behind studies (together with the DTMB 5415 appended hull). As a comparison of the CPP 1304 performance in OW and in-behind was planned from the start, the OW case was treated in two alternative ways, namely using a single domain rotating on the whole and two domains (one rotating cylindrical volume around propeller and hub and one large fixed volume) connected by sliding interfaces. In any case the grids around the blades were adjusted to resolve the laminar sub-layer in model scale, i.e. the cells at the wall show $y^+ ≈ 1$.

$K_T$ and $K_Q$ from the experiment were available with and w/o hub correction. As usual $K_T$ represents a normalized thrust defined as

$$K_T = T/(\rho n^2 D^4)$$ \hspace{1cm} (1)

The torque coefficient $K_Q$ normalises the Torque similarly

$$K_Q = Q/(\rho n^2 D^5)$$ \hspace{1cm} (2)

As the OW tables delivered from EFD usually isolate the blade forces we separated the blades also in the post processing of the CFD output. The $K_T$ -comparison of EFD and CFD on this basis is given in Figure 1 by the dashed lines. For the CPP 1304 also uncorrected measurements were available, which logically included blade and hub forces as a total. Adding the hub parts in the CFD post processing and doing the ‘Blades&Hub’-comparison gives the full lines in Figure 1, which reflect a much better agreement. The reason, why a hub correction performed in EFD - on the basis of an isolated hub run - may isolate the blade forces insufficiently is demonstrated with Figure 2. It gives a sample for a typical pressure field developing on the nose cap and entering the total thrust balance. In this view the blades have been removed to isolate the cap surface, but they actually trigger the pressure on large parts of the cap. The moments are hardly effected by geometrical details for the hub and a good agreement on $K_Q$ was already obtained on the basis of the ‘corrected’ EFD data.

![Figure 1: Thrust coefficient KT in EFD and CFD (FresCo+-results) for model scale, referencing on one hand only forces on blades (‘Blades’) and on the other hand blades and hub forces (‘Blades&Hub’).](image-url)
2.2 Propeller 1304 in hypothetic Open Water modes

It was planned to run a hypothetic model scale self-propulsion simulation where the propeller shows a ‘slip wall’ boundary condition while hull and appendages are treated ‘non-slip’ as usual. To process the related in-behind propeller performance data a similar hypothetic OW scenario was required. Figure 3 shows open water results calculated for model scale under ‘non-slip wall’ and ‘slip wall’ settings. Besides $K_T$ and $K_Q$ also the OW efficiency $\eta_o$ is given ($\eta_o = (T u)/(2\pi n Q)$, $u$ denoting the advance velocity). Figure 3 includes a true full scale case using dimension D=3 m and shaft frequency $n=4.33$ 1/s. These settings reflect the demands of a recent ITTC benchmark call on propeller scaling (based on CPP 1304).

According to Figure 3 we obtained nominally higher scale effects on thrust (thrust coefficient $K_T$) than on torque (torque coefficient $K_Q$) while the full-scale efficiency was behaving as expected. The ‘slip wall’ case represents a performance extreme, which roughly doubles the efficiency offset already existing between model and full scale, the latter case treated as hydraulically smooth here.

We also dealt with the hypothetic scenario of a reversed open water setup, which resembles the combination of blades, hub and cap from the propulsion mode (Figure 4). For the isolated blade forces and moments we found hardly any difference in this case.

![Figure 2: Pressure on nose cap of CPP 1304 in POW mode (flow from left to right)](image)

![Figure 3: Open Water calculations in different scale: ‘MS’ for Model Scale, ‘S’ for full scale and ‘SW’ for a slip wall condition set on the blade surface (FreSCo+-results)](image)
3 PROPELLER 1304 BEHIND HULL DTMB 5415

In order to compare results of different institutions the in-behind setup was prescribed in terms of model speeds, geometry of hull and appendages, propeller position as well as propeller blade- and hub-details. Grids were generated under the demand, that the laminar sub-layer on the propeller blade surface should be resolved ($y^+ \approx 1$). However the connection of the rotating propeller to the cylindrical strut barrel and the shaft was meshed differently. For the setup treated with Fluent a complete gap upstream of the propeller close to the blade root sections was introduced. For the scenario treated with FreSCo+ initially, a gapless connection was set at that point. The Fluent results on thrust showed a strong response to this mesh detail and so did the so-called ‘small figures’, in particular the ‘relative rotative efficiency’ $\eta_R$ and the wake fraction. We recall that $\eta_R$ represents the ratio of OW and in-behind torque coefficients at $K_T$-Identity:

$$\eta_R = \frac{K_q^{\text{OW}}}{K_q^{\text{BH}}}$$

To be comparable with the setup used for Fluent, the FreSCo+ analysis was recently also done with gap as well. In this analysis a centre shaft was added, closing the gap near the axis.

3.1 Numerical treatment of self-propulsion

Self-propulsion of the DTMB 5415 vessel was considered in model scale. Table 2 gives the main particulars for the hull in model and full scale. As already listed in Table 1 the combination of DTMB 5415 and propeller CPP 1304 was established under the assumption that the full scale diameter should read $D_{FS} = 5.1$ m.

In view of a numerical ‘British Method’ serving to find the self-propulsion point, two suitable shaft frequencies were to be estimated for every ship speed. To meet the actual self-propulsion point by linear interpolation, the lower estimate should represent a state with insufficient propeller thrust and the higher guess should resemble a state with too highly loaded propeller. Using the OW data of CPP 1304 and referencing results on thrust coefficient
KT and shaft frequency $n$ related to full scale propulsion of DTMB 5415 with another propeller [2] we arrived at the shaft frequency guesses given in Table 3. As model scale propulsion was simulated under the DB approach this table also includes offset forces. To define self-propulsion they are to be added to (resp. subtracted from) the numerical derived hull forces. These offsets consist of a friction deduction force $F_D$ derived by HSVA and an estimate of the pure wave resistance of the hull derived by two phase flow calculations on hull resistance using Fluent. According to Table 3 the self-propulsion point is related to a positive offset $F_{HULL} - T_{TOTAL}$ for Froude number $F_n = 0.28$ and to a negative offset $F_{HULL} - T_{TOTAL}$ at $F_n = 0.41$ ($T_{TOTAL}$ stands for the thrust of the twin screw system).

The grids generated for Fluent and FreSCo+ consisted of two domains connected by sliding interfaces, namely a rotating cylinder for the cells around propeller and hub and a fixed cell system for the remaining flow regime. We applied two alternatives for the effective treatment of the rotating propeller while the hull flow develops. The ‘frozen rotor’ and the rotor driven by a ‘ramp’-function (reducing initially large time steps continuously) were the applied techniques to reduce the computational time.

### Table 2: Particulars of DTMB 5415 in Full Scale and Model Scale ($\lambda = 24.825$)

<table>
<thead>
<tr>
<th></th>
<th>FS</th>
<th>MS</th>
</tr>
</thead>
<tbody>
<tr>
<td>$L_{pp}$ (m)</td>
<td>142</td>
<td>5.719</td>
</tr>
<tr>
<td>$L_{wl}$ (m)</td>
<td>142.18</td>
<td>5.726</td>
</tr>
<tr>
<td>$B_{wl}$ (m)</td>
<td>19.06</td>
<td>0.768</td>
</tr>
<tr>
<td>$T$ (m)</td>
<td>6.15</td>
<td>0.248</td>
</tr>
<tr>
<td>$S$ w/o rudder (m$^2$)</td>
<td>2972.6</td>
<td>4.823</td>
</tr>
</tbody>
</table>

### Table 3: Conditions for Model Scale propulsion ($F_n, V, n$), estimated performance of (one) propeller ($w_e, K_T, T_1$) and DB propulsion point in terms of $F_{HULL} - T_{TOTAL}$ (involving wave resistance $R_r$ and friction deduction $F_D$)

<table>
<thead>
<tr>
<th>$F_n$</th>
<th>$V$ [m/s]</th>
<th>$n$ [1/s]</th>
<th>$w_e$ (estimate)</th>
<th>$T_1$ in OW [N]</th>
<th>$K_T$</th>
<th>$F_D$ [N]</th>
<th>$R_r$ [N] (estimate)</th>
<th>$F_{HULL} - T_{TOTAL}$ = $F_D - R_r$ [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.28</td>
<td>2.094</td>
<td>7.288</td>
<td>0.04</td>
<td>1.342</td>
<td>193239</td>
<td>0.204</td>
<td>11.85</td>
<td>8.50</td>
</tr>
<tr>
<td>0.28</td>
<td>2.094</td>
<td>7.943</td>
<td>0.04</td>
<td>1.232</td>
<td>299294</td>
<td>0.266</td>
<td>11.85</td>
<td>8.50</td>
</tr>
<tr>
<td>0.41</td>
<td>3.071</td>
<td>11.724</td>
<td>0.04</td>
<td>1.224</td>
<td>661855</td>
<td>0.27</td>
<td>22.72</td>
<td>80.00</td>
</tr>
<tr>
<td>0.41</td>
<td>3.071</td>
<td>12.88</td>
<td>0.04</td>
<td>1.114</td>
<td>973363</td>
<td>0.329</td>
<td>22.72</td>
<td>80.00</td>
</tr>
</tbody>
</table>

### 3.2 Results on numerical self-propulsion

A first interesting finding from the common test case treatment is given by the history of the single blade forces as displayed in Figure 5 (showing higher fluctuations than expected). It was also interesting to check propeller forces in detail and investigate gap effects. The treatment of the gap between the hub and the strut barrel (modelled or not) will not influence the self-propulsion point (in view of the resulting shaft frequency) but it will affect the thrust dedicated to the propeller unit. All Fluent results on self-propulsion at the speeds and shaft frequencies covered by Table 3 were obtained with a complete gap between propeller hub and
shaft barrel (see Figure 6). The mid picture of this figure already indicates the negative pressure acting on the gap disc representing part of the propeller hub. When the self-propulsion point was deduced from the DB Fluent-calculations at Fn=0.28 and Fn=0.41 we used the forces offsets FHULL – TTOTAL given in Table 3. According to Figure 7 representing outward turning propellers at Fn=0.41 the ‘British Method equivalent’ self-propulsion point was obtained at T1=93.04 N. Note, that in this figure only half of the offset force is plotted on the horizontal axis as only one propeller is referenced.

**Figure 5:** DB Fluent-calculations for outward turning at n=11.72 [1/s]; generating an equivalent single blade thrust history by analysing individual blade thrusts at one time step

**Figure 6:** DB Fluent-calculations addressing gap effects at outward turning twin-screw propeller at Fn=0.41: pressure on hull/propeller without gap (left) and with gap (right)

**Figure 7:** DB Fluent-calculations for outward turning at Fn=0.41: evaluation of propeller thrust at self-propulsion via linear interpolation in a ‘thrust over (hull force minus thrust)’-diagram following the ‘British method’ known from towing tank experiments.
The FreSCo\textsuperscript{+}-calculations did not include the gap initially. The analysis with FreSCo\textsuperscript{+} was then repeated on a grid which shows a 3 mm gap in model scale (HSVA’s standard for propulsion test setups). To comply with test setups, the gap was closed by a shaft dummy in the vicinity of the shaft axis. According to Figure 8 the DB FresCo\textsuperscript{+}-calculations hardly show any global difference in the pressure on hull, shaft and rudder when comparing without gap (left picture) and with gap (right picture). The detailed visualization done in Figure 9 confirms, that the pressure in the gap ranges on a quite constant and negative level, as already recognized for the Fluent results. In return, a corresponding low pressure has also been detected for the opposite gap disc belonging to the ship. Consequently the force balance for the whole system does not change with or without gap, neither does the self-propulsion point.

3.3 Further processing of data from numerical self-propulsion

Depending on the shaft frequency setting, we noticed an increase of the propeller thrust by about 3-7\% when the force on the gap is added. By including the gap and following the model test evaluation procedure, we typically identify an increase in ‘relative rotative efficiency’ $\eta_R$ by about 4 \%. Figure 10 gives an example on the $\eta_R$-results for an outward turning case at $F_n=0.41$ and $n_2=12.10$ 1/s (interpolated self-propulsion point from FreSCo\textsuperscript{+}-results). According to this figure the lowest $\eta_R$-value is obtained when neglecting of the gap force and including the forces on the downstream cap. The $\eta_R$-value from ‘blades only’ ranges slightly higher than the latter. Note, that in any case the ‘blades only’ $K_T$ and $K_Q$ were referenced in the computed model scale OW results.

As announced above an in-behind analysis was done using FreSCo\textsuperscript{+} giving the propeller blades a ‘slip wall’ boundary condition. For the outward turning case at $F_n=0.41$ the Figure 11 shows $\eta_R$-results obtained at the two shaft frequencies either combining ‘slip wall’ propulsion with ‘slip wall’ OW or ‘regular’ propulsion with ‘regular’ OW. The artificial ‘slip wall’ setting does not change the $\eta_R$-results too much (the isolated dependence on the shaft frequency shows similar differences).

4 CONCLUSIONS

Summarizing one may state that more strictly than initially expected the system of blades and hub must be taken as a unit and geometrical details around the hub must be carefully reflected to allow for a true comparison of EFD and CFD. For model scale propulsion as well as for OW the grids around the blades were adjusted to resolve the laminar sub-layer. This high resolution request is connected to superior efforts in grid generation and to probably enlarged computation time. Depending on the concerns – above we put some stress on $\eta_R$ - the ‘slip wall’ setting may represent a chance to ease the numerical propulsion simulations.

ACKNOWLEDGEMENTS

This work is linked to the INRETRO project realized as a European ERA-NET venture in the MARTEC framework. The financial support by the national funding associations is gratefully acknowledged.
Figure 8: DB FresCo\textsuperscript{+}-calculations addressing gap effects at outward turning twin-screw propeller at \( F_n=0.41 \): pressure on hull/propeller without gap (left) and with gap (right).

Figure 9: FresCo\textsuperscript{+}-calculations with resolved gap for propeller in propulsion mode indicating negative pressure in the gap.

Figure 10: DB FreSCo\textsuperscript{+}-calculations in model scale, outward turning with \( n=12.10 \text{ 1/s} \) (interpolated self-propulsion point at \( F_n=0.41 \)): evaluation of \( \eta_R \) with and without reference to hub-parts (in the latter case either ‘Hub Cap & Gap’ or ‘Hub Cap only’).
Figure 11: DB FreSCo+-calculations in model scale for outward turning with $n_1=11.72$ [1/s] resp. with $n_2=12.88$ [1/s] (Fn=0.41): evaluation of $\eta_R$ without reference to hub-parts using either the combination ‘non-slip wall OW’&‘non-slip wall behind’ or ‘slip wall OW’ & ‘slip wall behind’

REFERENCES


MARINE CYCLOIDAL PROPULSION MODELLING FOR DP APPLICATIONS

MARCO ALTOSOLE*, SILVIA DONNARUMMA†, VALENTINA SPAGNOLO‡ AND STEFANO VIGNOLO‡

* Department of Electrical, Electronic, Telecommunications Engineering and Naval Architecture (DITEN)
Polytechnic School of Genoa University, Italy
Via all’Opera Pia 11A, 16145 Genova, Italy
e-mail: marco.altosole@unige.it

‡ Department of Mechanical, Energy, Management and Transport Engineering (DIME)
Polytechnic School of Genoa University
Via all’Opera Pia 15, 16145 Genova, Italy
e-mail: spagnolo.v86@gmail.it, vignolo@dime.unige.it

† Department of Industrial Engineering (DII)
University of Trento
Via Sommarive 9, 38123 Povo (TN)
e-mail: silvia.donnarumma@unitn.it

Key words: cycloidal propeller, dynamic positioning systems, simulation

Abstract. This paper presents the numerical modelling of a cycloidal propeller in free-running conditions together with its possible applications. The model calibration is carried out by comparing simulation results with experimental data of an existing cycloidal unit. The achieved results support the main strength of the proposed simulation approach: propeller fluid dynamics is not calculated, avoiding demanding computations that would not allow an effective simulation of the whole propulsion plant. As a case study, the cycloidal propulsors model is used for the thruster allocation assessment of the Dynamic Positioning (DP) system of a surface vessel, originally equipped with traditional propellers. Then, the steady-state performance analysis of the DP system is carried out in terms of a comparison between the two distinct propulsion configurations.

1 INTRODUCTION

Cycloidal propellers are able to provide thrust by rotating and additionally oscillating blades. They are classified into three main types: true cycloidal (e.g. Kirstem Boeing Propeller), epicycloidal (e.g. Voith Schneider Propeller) and trochooidal propellers (e.g. Whale Tail Wheel Propeller). The different types of cycloidal propellers are defined by their eccentricity value $e$, i.e. the ratio between the distance of the steering center from the propeller axis, and the radius of the orbit which defines the position of the blade axes. According to this definition [1], an epicycloidal propeller has $e < 1$, a true cycloidal propeller is characterized by $e = 1$ and a throcoidal propeller has $e > 1$. 
The numerical modelling described in the present study is referred to epicycloidal propulsors. The propeller thrust and torque modelling is based on the kinematical aspects of the blades motion, taking into account suitable correction factors in order to properly consider “dissipative” phenomena such as: interference between blades, the shielding induced by the half of the rotor which receives the incoming flow, and the slight reduction of the reverse thrust. The calibration of the simulator is carried out by comparing simulation results with experimental ones, pertinent to an existing cycloidal unit. The main strength of the proposed simulation approach is from not having to calculate the propeller fluid dynamics, avoiding demanding computations that would not allow an effective simulation of the whole propulsion plant. CFD codes are usually useful for the blade design and then they are mainly used by manufacturers (e.g. a RANS equation code has been used by Voith Schneider to represent the behavior of their epicycloidal propellers [2]), while for other applications simplified approaches can be often more suitable, because able to represent the overall performance of the propeller starting from a very few input data and with a reduced computation time. Similar performance prediction methods are commonly used for traditional marine propellers (e.g. systematic propeller series) and waterjets [3]. On the contrary, in the case of cycloidal propellers, manufacturers do not publicly share their performance maps for confidential reasons, and then simplified simulation approaches are more difficult to be developed. The present simulation method is based on a mixture of theoretical and empirical considerations, as already proposed, although in a very different way, by Taniguchi [4]. In the latter, the total thrust and torque of the propeller are evaluated by integrating the lift and drag forces acting on each blade section and a correction factor is introduced to consider non-uniformity of induced velocity over the blade length.

As an application of the proposed method, the authors discuss different thrust allocation logics of a dynamic positioning (DP) system for a surface vessel, equipped with two cycloidal propellers and a single bow thruster. The examined ship is the same for which a DP system, characterized by a conventional twin-screw propulsion, was already developed and installed on board [5].

2 CYCLOIDAL PROPELLER KINEMATICS

2.1 Kinematical model

In this section, the kinematical model describing the motion of each blade of the cycloidal propeller is presented. For simplicity, a two dimensional plane model is adopted, where two distinguished reference frames are introduced: the first one $\{O, \mathbf{b}_1, \mathbf{b}_2, \mathbf{b}_3\}$ is fixed to the hull and it has its origin $O$ at the center of the rotor, the unit vector $\mathbf{b}_1$ points towards the bow, the unit vector $\mathbf{b}_2$ points towards starboard and the unit vector $\mathbf{b}_3 = \mathbf{b}_2 \times \mathbf{b}_1$ points downwards; the second one $\{O, \mathbf{e}_1, \mathbf{e}_2, \mathbf{e}_3\}$ rotates clockwise about the vertical axis passing through $O$ and parallel to $\mathbf{b}_3 = \mathbf{e}_3$, by an angle $\beta \in [0,2\pi]$ which determines (the perpendicular of) the steering force direction. The angle $\beta$ is then related to the rudder pitch of the cycloidal propeller. The steering center $C$ lies on the straight line passing through $O$ and parallel to $\mathbf{e}_2$. The linear transformation between the bases $\{\mathbf{b}_1\}$ and $\{\mathbf{e}_1\}$ is expressed as
During the revolution motion, the projection \( P \) of the blade shaft on the plane \( \langle O, b_1, b_2 \rangle \) describes a circumference having center \( O \) and radius \( R \) coinciding with the rotor radius. In Cartesian coordinates associated with the frame \( (O, b_1, b_2, b_3) \), such a circumference is parameterized by

\[
P(\theta) : \begin{cases} x = R \cos \theta \\ y = R \sin \theta \\ z = 0 \end{cases}
\]

where \( \theta \) denotes the angle (function of time) describing the revolution motion of the blade. In the vector basis \( \{b_i\} \), the unit vector \( t \) tangent to the circular path of \( P \) has components of the form

\[
t(\theta) = \begin{cases} t_1 = -\sin \theta \\ t_2 = +\cos \theta \\ t_3 = 0 \end{cases}
\]

Introducing the vector

\[
(C - O) = s e_2 = -s \sin \beta \ b_1 + s \cos \beta \ b_2 \quad s \in [0, 0.8R]
\]

the vector joining the steering centre \( C \) with the point \( P \) can be expressed as

\[
(P - C) = (R \cos \theta + s \sin \beta) b_3 + (R \sin \theta - s \cos \beta) b_2
\]

The variable \( s \) is usually called driving pitch and it controls thrust magnitude. The unit vector orthogonal to \( (P - C) \) and belonging to the plane \( \langle O, b_1, b_2 \rangle \) identifies with the unit vector of the blade chord and it is given by

\[
\quad \frac{(P - C)^\perp}{|(P - C)^\perp|} = \frac{(-R \sin \theta + s \cos \beta) b_2 + (R \cos \theta - s \sin \beta) b_2}{\sqrt{(-R \sin \theta + s \cos \beta)^2 + (R \cos \theta + s \sin \beta)^2}}
\]

The pivoting motion of the blade around its shaft can be characterized by the angle \( \alpha \) (function of time) between the unit vectors \( \vec{t} \) and \( \frac{(P - C)^\perp}{|(P - C)^\perp|} \). Due to the relation

\[
cos \alpha = \frac{(P - C)^\perp}{|(P - C)^\perp|} \cdot \vec{t} = \frac{R + s \sin (\beta - \theta)}{\sqrt{(-R \sin \theta + s \cos \beta)^2 + (R \cos \theta + s \sin \beta)^2}}
\]

where the dot denotes the usual scalar product between vectors, the pivoting angle \( \alpha \) can be defined as
choosing anticlockwise the positive direction of rotation around the blade shaft. The above outlined kinematical model can be summarized by Figure 1.

Figure 1: Kinematics of the blade

Supposing now that the vessel is moving, let \( \mathbf{v}_O = \hat{u} b_3 + \hat{\beta} b_2 \) be the velocity of \( O \) (w.r.t. the Earth-fixed frame) expressed in the hull-fixed basis. Denoting by \( \mathbf{v}_P' = -R \hat{\theta} \sin \theta \ b_3 + R \hat{\theta} \cos \theta \ b_2 \) the velocity of the point \( P \) w.r.t. the body-fixed frame, the velocity of \( P \) w.r.t. the Earth-fixed frame is given by

\[
\mathbf{v}_P = \mathbf{v}_P' + \omega \wedge (P-O) = [\hat{u} - R(\hat{\theta} + r) \sin \theta] b_3 + [\hat{\beta} + R(\hat{\theta} + r) \cos \theta] b_2
\]

where \( \omega = r b_3 \) is the angular velocity of the vessel. The velocity of the incoming flow experienced at \( P \) by a blade-fixed observer is, then \(-\mathbf{v}_P\); its unit vector \( \hat{\ell} \) is expressed as

\[
\hat{\ell} = -\frac{\mathbf{v}_P}{|\mathbf{v}_P|} = \frac{[\hat{u} - R(\hat{\theta} + r) \sin \theta] b_3 + [\hat{\beta} + R(\hat{\theta} + r) \cos \theta] b_2}{\sqrt{[\hat{u} - R(\hat{\theta} + r) \sin \theta]^2 + [\hat{\beta} + R(\hat{\theta} + r) \cos \theta]^2}}
\]

Making use of the unit vector \( \hat{\ell} \), it is possible to characterize the attack angle of the incident flow as

\[
\hat{\alpha} = \pi - \cos^{-1} \left[ \frac{(P - C) \perp}{|(P - C) \perp|} \cdot \hat{\ell} \right]
\]
2.2 Hydrodynamic forces

In this section, making use of some simplifying assumptions, a suitable model for evaluating the hydrodynamic forces generated by each blade is presented. It is supposed that the velocity of the incident flow be the same over the entire surface of the blade and coincide with $-v_p$. Under such a condition, the lift and drag produced by each blade can be expressed as

$$
L = c_L \frac{1}{2} \rho_w A |v_p|^2 \hat{n} \quad D = c_D \frac{1}{2} \rho_w A |v_p|^2 \hat{\epsilon}
$$

where $c_L$ is the lift coefficient, $c_D$ is the drag coefficient, $\rho_w$ is sea water density, $A$ is the blade lateral area, $|v_p|$ is the incoming flow speed, $\hat{\epsilon}$ is the unit vector of the lift force (unit vector of the incoming flow at $P$), and $\hat{n}$ is the unit vector of the drag force (perpendicular to $\hat{\epsilon}$).

In order to determine the unit vector $\hat{n}$, the following procedure is adopted in which two main scenarios are distinguished:

- the attack angle $\alpha$ belongs to the interval $[0, \frac{\pi}{2}]$, namely the incoming flow hits the blade from the front. In such a circumstance, the unit vector $\hat{n}$ is determined according to the requirements:

$$
\hat{n} = \begin{cases} 
    b_3 \land \hat{\epsilon} & \text{when } \hat{\epsilon} \land \frac{(P - C)^\perp}{|(P - C)^\perp|} \cdot b_3 > 0 \\
    -b_3 \land \hat{\epsilon} & \text{when } \hat{\epsilon} \land \frac{(P - C)^\perp}{|(P - C)^\perp|} \cdot b_3 < 0
\end{cases}
$$

- $\alpha \in \left[\frac{\pi}{2}, \pi\right]$, the incoming flow hits the blade from the back. In this case, $\hat{n}$ is singled out by the requests:

$$
\hat{n} = \begin{cases} 
    -b_3 \land \hat{\epsilon} & \text{when } \hat{\epsilon} \land \frac{(P - C)^\perp}{|(P - C)^\perp|} \cdot b_3 > 0 \\
    b_3 \land \hat{\epsilon} & \text{when } \hat{\epsilon} \land \frac{(P - C)^\perp}{|(P - C)^\perp|} \cdot b_3 < 0
\end{cases}
$$

As remaining particular cases, if $\alpha = 0$ or $\alpha = \pi$ there is no lift while if $\alpha = \frac{\pi}{2}$ then $\hat{n} = \hat{\epsilon}$.

The above described procedure allows to determine the lift and drag provided by each single
blade. The resultant hydrodynamic force generated by the cycloidal propeller can be computed as the sum of all contributions by each blade.

2.3 Torque acting on the rotor

The calculation of the torque acting on the rotor deserves a specific discussion. In order to accomplish this task, the Newton-Euler moment equation for each single blade has to be considered. Developed in the hull-fixed reference frame and with respect to the point $O$, the Newton-Euler moments equation for each blade can be expressed as

$$M^E_D + M^H_D + M^G_D + M^R_D + M^I_D = I_G(\omega) + \omega \wedge I_G(\omega) + m(G - O) \wedge \alpha_G$$  \hspace{1cm} (15)

where $M^E_D$, $M^H_D$, $M^G_D$, $M^R_D$, and $M^I_D$ are the engine, hydrodynamic, weight force, reactive force, and inertial force torques w.r.t. $O$, respectively; $I_G$ is the inertia tensor w.r.t. the center gravity $G$ of the blade; $\omega$ is the blade angular velocity w.r.t. the hull-fixed frame; $\alpha_G$ is the acceleration of $G$ w.r.t. the hull-fixed frame; and $m$ is the blade mass.

Knowing the revolution velocity of the rotor and the position of the steering center as well as the velocity of the incoming flow, the consequent motion of the blade is known from kinematics; at the same time, the knowledge of the hydrodynamic forces allows the evaluation of their moment. Neglecting the rolling friction around the rotation axis, the (scalar value of) engine torque amounts to the projection

$$M^E_D = -M^H_D \cdot b_3 - M^I_D \cdot b_3 + I_G(\omega) \cdot b_3 + m(G - O) \wedge \alpha_G \cdot b_3$$  \hspace{1cm} (16)

Once again, by adding all the contributions by each blade, the total engine torque can be obtained.

3 NUMERICAL MODELLING AND VALIDATION

The kinematical model has been used to develop a Matlab-Simulink simulator for cycloidal propellers. In this section, the main features and the validation of such simulator are presented.

3.1 Simplifying assumptions

In order to simplify the simulation platform, some hypotheses have been assumed:
- the propeller is considered in free-running conditions (no hull interference), as in open water, invested from an incoming flow;
- the problem is supposed to be stationary;
- the model is 2D;
- the contributions of each single blade are separately calculated and then summed. The interference among the blades is taken into account by means of correction factors.

3.2 Input data

The Simulink model needs input data, given from Matlab file. These data are: the geometry of the propeller (length, chord and orbit diameter of the blade – see Table 1); the sea water
characteristics (viscosity and density); the $C_L$ and $C_D$ coefficients of the blade (obtained by a previous CFD calculation [6]); the rotor speed and maximum pitch available; the steering pitch angle ($0^\circ$ in forward direction, $180^\circ$ in astern condition) and the driving pitch (expressed as a percentage of the radius of the orbit of the blades).

<table>
<thead>
<tr>
<th>Table 1: Propeller geometric parameters.</th>
</tr>
</thead>
<tbody>
<tr>
<td>N. of blades</td>
</tr>
<tr>
<td>Diameter (m)</td>
</tr>
<tr>
<td>Blade length (m)</td>
</tr>
<tr>
<td>Blade Chord (m)</td>
</tr>
<tr>
<td>Max tip thickness</td>
</tr>
</tbody>
</table>

### 3.3 Simulation

The simulator allows to generate the curves of the coefficients $K_S$ and $K_D$, respectively defined by

$$K_S = \frac{T}{\frac{1}{2} \rho w D L u^2}$$  \hspace{1cm} (17.a)

$$K_D = \frac{4M}{\rho w D^2 L u^2}$$  \hspace{1cm} (17.b)

and depending on the advance coefficient

$$\lambda = \frac{V_A}{\pi n D}$$  \hspace{1cm} (17.c)

where $T$ is the total thrust, $M$ is the total torque acting on the rotor, $L$ is the blade height, and $u = \pi n D$ is the revolution speed of the blades.

The whole model consists of a set of identical subsystems, each of them representing the behaviour of a single blade. Making use of Eqs. (12) and (15), the components of total thrust and torque are calculated in the basis $\{ b_i \}$. According to Eqs. (17.a) and (17.b), it is then possible to evaluate the coefficients $K_S$ and $K_D$. The simulation of coefficient $K_S$ and $K_D$ has been performed in the pitch range from 40% to 80%, with steps of 10%. In particular, Figure 3 shows the comparison between literature data (pertinent to a real existing cycloidal propeller, characterized by the same geometry) and the simulation of the coefficients $K_S$ and $K_D$ without any corrections for a pitch of 80%.
3.4 Model validation

The differences shown in Figure 3 are mainly due to the stated simplifying assumptions about the interactions among the blades. In order to minimize these differences, two correction factors have been introduced, taking two main phenomena into account: the shielding of the blades that are in the half circumference not directly exposed to the incoming flow and the interference of a blade with the other. The calibration of such correction factors has been carried out by comparing simulation results with the performance data of the existing propeller. The comparison is shown in Figure 4.

Once the curves of $K_S$ and $K_D$ have been obtained, the total thrust for an advance coefficient of 0.4 and different thrust directions has been analysed. The numerical modelling showed a slight reduction of thrust magnitude for the reverse thrust. In order to consider this further phenomenon, another correction factor has been introduced, according to the existing propeller performance. The results are illustrated in Figure 5.
Figure 4: Thrust and torque coefficients of the cycloidal thruster.

Figure 5: Thrust and torque coefficients of the cycloidal thruster.
Summarizing, the three considered corrections are:
- “Shielding correction”, referring to the shielding of the blades that are in the half circumference not directly exposed to the incoming flow: in the model, the correction factor, depending on driving pitch values, reduces the velocity of the incoming water flow.
- “Interference correction”: the interference among the blades is modelled by reducing the attack angle of the incoming flow in respect to the chord of the blade section; the correction depends on $\lambda$ and pitch values.
- “Reverse thrust correction”, representing the reduction of the reverse thrust.

The correction factors, empirically estimated for a particular propeller, are represented in terms of percentage reductions. In the case of cycloidal propellers with a different number of blades, the same correction values can be considered for an overall estimation of the propeller performance, achieving good results [7]; obviously, for a more reliable simulation, the correction factors have to be calibrated again for each particular application.

As it has been already mentioned, the main strength of the proposed simulation approach comes from not having to calculate the propeller fluid dynamics, avoiding demanding computations that would not allow an effective simulation of the whole propulsion plant.

As an application, the numerical model has been used for the performance evaluation of a DP system of a surface vessel equipped with two cycloidal propellers at the stern and a single bow thruster. The examined ship is the same for which a DP system, characterized by a conventional twin-screw propulsion, was already developed by the automation provider Seastema S.p.A. in cooperation with University of Genoa [5].

4 DYNAMIC POSITIONING APPLICATION

The first step in the assessment of the performance of a given dynamic positioning system is the evaluation of the static capabilities of the vessel. In order to accomplish this task, dynamic positioning polar plots are a useful tool. In the following, the DP polar plots for the vessel considered in our study are presented, taking two different allocation logics into account. Finally, comparisons with the DP capability of the real ship (equipped with a standard propulsion configuration) are reported.

4.1 Environmental disturbances

The analysis is performed in static conditions and involves the balancing of the forces and moments generated by environmental disturbances (selected from the world wide sea state table). Environmental disturbances are described as the sum of forces and moments due to wind, current and wave respectively. Forces and moments are expressed making use of the well-known resistance form, depending on non-dimensional coefficients $C_X(\gamma_r)$, $C_Y(\gamma_r)$, and $C_M(\gamma_r)$, related respectively to the longitudinal force, the lateral force and the moment. $\gamma_r$ is the relative angle between the disturbance incoming direction and the vessel heading. In order to consider the occurring worst condition, all environmental disturbances are supposed to be aligned in the same incoming direction, thus $\gamma_r$ is the same for current, wind and waves. The
current and the wind speeds are assumed constant and wave drift forces are modelled as proportional to the square of the significant height $H_s$ [5]. Collecting all the (b-basis) components of the force and moment in a unique 3-dimensional array $\tau$, we have

$$
\tau_E = \tau_{\text{current}}(\gamma_r) + \tau_{\text{waves}}(\gamma_r, H_s) + \tau_{\text{wind}}(\gamma_r, v_G)
$$

(18)

4.2 Thrust allocation logic

In order to assess the DP capability of the propulsion configuration with cycloidal propellers, two allocation logics have been developed and compared, with the one implemented on the real vessel, equipped with two traditional propellers and rudders. Details of the original allocation can be found in [5].

The first allocation logic is based on a constrained minimum problem. The idea is to minimize a cost function of seven variables $\mathbf{x} = [T_{pt}, T_{sb}, X_{bow}, Y_{pt}, X_{sb}, Y_{sb}] \in \mathbb{R}^7$, subject to some suitable constraints. In particular, defining by $T_{pt}$ and $T_{sb}$ respectively, the portside and starboard cycloidal propeller thrusts, $T_{bow}$ the thrust of the bow thruster, $(X_{pt}, Y_{pt})$ and $(X_{sb}, Y_{sb})$ the components of the portside and starboard thrust forces, the constrained minimum problem is formulated as

$$
\min_{\mathbf{x}} f(\mathbf{x}) \quad \text{with} \quad h_i(\mathbf{x}) = 0 \quad \text{and} \quad g_j(\mathbf{x}) > 0
$$

(19)

where

$$
f(\mathbf{x}) = \left( \frac{T_{pt}}{T_{max}} \right)^2 + \left( \frac{T_{sb}}{T_{max}} \right)^2 + \left( \frac{T_{bow}}{T_{max}} \right)^2
$$

(20)

is the cost function to be minimized, and

$$
\begin{align*}
    h_1(\mathbf{x}) &= X_{env} - X_{pt} - X_{sb} \\
    h_2(\mathbf{x}) &= Y_{env} - Y_{pt} - Y_{sb} - T_{bow} \\
    h_3(\mathbf{x}) &= N_{env} - x_{bow} T_{bow} - x_{pt} Y_{pt} + y_{pt} X_{pt} - x_{sb} Y_{sb} + y_{sb} X_{sb} \\
    h_4(\mathbf{x}) &= T_{pt}^2 - X_{pt}^2 - Y_{pt}^2 \\
    h_5(\mathbf{x}) &= T_{sb}^2 - X_{sb}^2 - Y_{sb}^2
\end{align*}
$$

(21)

are the constraints to be satisfied. In Eq. (21) $\{X_{env}, Y_{env}, N_{env}\}$ are the components of the force and the moment due to environmental disturbances, $(x_{pt}, y_{pt})$ and $(x_{sb}, y_{sb})$ are the coordinates of the propellers thrust centres and $x_{bow}$ is the longitudinal coordinate of the bow thruster. Moreover, we have

$$
g_1(\mathbf{x}) = x(2) \quad , \quad g_2(\mathbf{x}) = x(3)
$$

(22)

Eq. (19) requires that the sum of the squared desired thrusts is minimum. Eq.(21) details the constraints: the first three represent the static equilibrium between the environmental disturbances and the delivered force and moment; the last two correlate the portside and
starboard thrust force with their longitudinal and lateral components. Finally, Eq. (22) ensures that the modulus of the two aft thrusts is positive.

The second allocation algorithm is based on the idea that one cycloidal propeller is designed to compensate the environmental resultant force, while the other one together with the bow thruster, is devoted to compensate the moment. This allocation configuration is supposed to be more stable when environmental disturbances are relatively small and constant in time. In this case, the force and moment balance is uniquely determined in an algebraic way, whenever the thruster devoted to the force compensation is chosen. The choice of such a thruster relies on the requirement that the moment generated by the thruster itself has opposite sign with respect to the disturbances one. More explicitly, we have

\[
T_i = \sqrt{X_{env}^2 + Y_{env}^2}, \quad \delta_i = \tan^{-1} \frac{Y_{env}}{X_{env}}
\]

\[
T_{bow} = -T_j
\]

\[
T_j = \frac{x_i T_i \sin \delta_i - y_i T_i \cos \delta_i + N_{env}}{x_j - x_{bow}}
\]

where \(i = pt\) and \(j = sb\) when \(|N_{pt} - N_{env}| > |N_{sb} - N_{env}|\) and \(i = sb\) and \(j = pt\) otherwise; \(N_{pt}\) is the moment generated by the propeller if portside is the one compensating the force and viceversa for \(N_{sb}\).

The criterion adopted to choose the thrust devoted to compensate the disturbances force is based on computing, for both the possible choices, the moment generated by the thruster and adding it to the environmental one. Such resulting moments are compared and the thruster generating the minimum moment is the one chosen for the environmental force compensation.

4.3 Results

The station keeping capability of a dynamically positioned vessel is often presented by means of polar plots, which illustrate the steady-state performance of the vessel under certain environmental conditions. A capability plot shows the maximum weather conditions in which the vessel can maintain its position and heading, obeying upon a certain thrust allocation logic. DP capability plots are drawn assuming that the all environmental disturbances come from the same direction. In this case study, the vessel is supposed to operate in Mediterranean Sea with a significant wave height \(H_s\) of 2.5\,m and a constant current speed \(v_c\) of 1\,kn. Instead, the wind speed keeps increasing until the vessel is able to sustain the wind load, namely until the resultant environmental forces and moments are balanced by the maximum available thrust. Assuming that the aligned environmental disturbances rotate around the vessel (anticlockwise starting from zero which corresponds to stern), in Figure 6 the intersection of the curve with the radius of the circumference indicates the maximum wind speed at which the vessel can maintain its position and heading. For reasons of readability of the plot, a saturation was added when the maximum wind speed was higher than 60\,kn. Moreover, higher wind conditions involves sea disturbances that cannot be studied in the static case.
As it can be seen in Figure 6, maximum values of wind speed can be reached for disturbances coming from bow directions, while we have lower values for beam and stern directions. That is due to the larger areas exposed to the environmental loads. Regarding the differences due to the distinct propulsion configurations, numerical results show that the conventional propulsion layout is able to ensure only a limited DP capability (the yellow line), while great improvements could come from the adoption of cycloidal propellers in the propulsion system. Concerning the different allocation logics adopted for cycloidal propellers, it is evident that the optimization of the thrust (the red line) gives the best results, though this is mainly true for bow and quartering sea states, while for stern sea states the two allocation logics seem to give closer results.

Figure 6: DP capability polar plots for different thrust allocation logics.

8 CONCLUSIONS

A simulation model for ship cycloidal propellers has been presented. The simulator has been calibrated by comparing simulation data with experimental ones. The mathematical and numerical modelling of the free running behaviour of cycloidal propellers, in different operating conditions, is described without using a proper - but also demanding - fluid dynamics computation (CFD method is only used for the evaluation of the lift and drag coefficients of the single blade).
The core of the model is represented by the kinematical description and by the empirical correction factors that can be used for a preliminary performance estimation of several cycloidal propulsion units, characterized by different lengths and number of blades.

In this light, the performance analysis of the DP application shown in this study can represent a possible proper application of the developed propulsion model. In particular, a comparison between a traditional propulsion system and the cycloidal one has been carried out in terms of DP capability plots. Further considerations, in terms of comparison, could be made also in case of different cycloidal units, by changing blades area and number.

The present work aims to be the first step towards the implementation of a numerical model for the dynamical simulation of the manoeuvrability, at both low and high velocities, of vessels equipped with cycloidal propellers.

REFERENCES
ADJOINT-BASED OPTIMIZATION METHODS FOR FLOW PROBLEMS

Thorsten Grahs* and Johann Turnow†

* Institute of Scientific Computing
  Technische Universität Braunschweig
  Mühlenpfeldstrasse 23, 38106 Braunschweig, Germany
  e-mail: t.grahs@tu-bs.de

† Chair of Modelling and Simulation
  University of Rostock
  Albert-Einstein-Str. 2, 8059 Rostock, Germany
  email: johann.turnow@uni-rostock.de

Key words: Computational Methods, Optimization, Adjoint, Sensitivity Analysis

Abstract. Over the last decade, adjoint sensitivity analysis has become an established technique for the task of shape optimisation when many degrees of freedom are present. The success stems from the fact that the adjoint approach only needs one flow simulation for both the primal and the adjoint system, no matter how many design parameters are present. The derivation of the continuous adjoint approach is based on an augmented cost function which inheres the primal governing equations (here the RANS-equations) as constraints which have to be satisfied in the computational domain. Accordingly, the primal RANS equations are augmented with Lagrange multipliers and added to the thermal-fluid dynamic cost function. For shape optimisation, the variational formulation of the augmented cost function indicates the behaviour of the cost function with the variation of the shape, i.e. the variation of the surface mesh in normal direction.

We present the derivation and application of the continuous adjoint approach for the incompressible Reynolds-averaged Navier-Stokes (RANS) equations augmented with heat transfer. The derived approach is implemented into the framework of the C++ CFD toolbox OpenFOAM in order to derive a complete design cycle for shape optimisation. The derived optimisation process is applied to dimpled surface geometries in order to optimise cooling devices.

1 INTRODUCTION

Originally arising in control theory [1], adjoint sensitivity analysis has made its way into the area of fluid mechanics [2, 3]. Since then the method has become an established technique for shape and topology optimisation of fluid problems, especially when many degrees of freedom are present [4, 5]. The success stems from the fact that the adjoint approach only needs two flow simulations, one for the primal and one for the adjoint system, no matter how many design parameters are present. This is a clear advantage over standard parametric geometry
optimization, which needs usually as many flow solutions as parameter combinations are present. This benefit, especially for large application cases and consequently adjoint based optimization methods, has become an important tool in the optimization of industrially relevant application (e.g. [6–12]).

In this approach, we apply the continuous adjoint approach to the incompressible Reynolds-averaged Navier-Stokes (RANS) equations augmented with heat transfer. We derive the according adjoint system and adjoint boundary conditions for maximizing heat flux over a certain boundary. The derived optimization process is applied to dimpled surface geometries (see [15,16]) in order to optimize cooling devices.

2 THE GENERAL OPTIMIZATION PROBLEM

Let $I$ be a specific cost function defined on an admissible domain $\Omega \subset \mathbb{R}^N$ with boundary $\Gamma$. The domain $\Omega$ will be allowed to vary during the design process and is parametrized through a set of design variables $\beta$. In addition, a set of given constraints $r(s)$ has to be obeyed, typically a set of partial differential equations (governing equations) with state variables $s$. We can formulate the problem by

$$\max_{\beta} I(s, \beta) \text{ subject to } r(s, \beta) = 0 \text{ in } \Omega.$$  \hspace{1cm} (1)

This means we adapt the domain $\Omega$ by changing the design parameter $\beta$ in order to improve the cost function $I$.

The dependency of the cost function $I(s, \beta)$ with respect to their parameters is expressed by their total variation:

$$\delta I = \delta_I = \delta_{s} I + \delta_{\beta} I = \frac{\partial I}{\partial s} \delta s + \frac{\partial I}{\partial \beta} \delta \beta.$$

The necessary information for geometry variation comes from the so-called sensitivity $\partial I/\partial \beta$ of the cost function with respect to the design parameters. The sensitivity reveals how the cost function is affected by an admissible variation $\delta \beta$ of the design parameters $\beta$. In our approach, we assume shape optimization, which means we want to deform the surface mesh of the simulation domain. Thus, the design parameters are the positions of the nodes of the surface mesh. Variation of the design parameter means in this context the variation in the direction of the corresponding normal vector, i.e. inward or outward movement of the surface node.

In general, the evaluation of the second right hand side term of (2) requires a solution of the flow field for each design parameter $\beta_i$ [4,5]. Thus, if one considers the variation of the surface nodes as design parameters, as we are going to do in the following, one has to carry out as many solution $r(s, \beta_i)$ as design parameters $\beta_i$ are present. Typically, the number of surface nodes $\beta_i$ is of the order $i = 10^6 - 10^7$. In order to avoid such computational effort, on changes from to the dual or adjoint formulation of the governing equations, what we demonstrate in the next sections.
3 THE PRIMAL EQUATION SYSTEM

The primal equation system of the optimization problem is constituted by the governing
equations of the flow problem, i.e. the incompressible Navier-Stokes equations:

\[ \begin{align*}
\partial_t (\rho \mathbf{u}) + (\mathbf{u} \cdot \nabla) \mathbf{u} &= -\nabla p + \nabla \cdot [2\nu \mathbf{D(u)}], \\
\partial_t \rho + \nabla \cdot \mathbf{u} &= 0.
\end{align*} \quad (3) \]

This equations forms the primal system with primal variables: pressure \( p \) and velocity \( \mathbf{u} \) = \((u_1, \ldots, u_3)^T\). Here, \( \mathbf{D(u)} = \frac{1}{2}[\nabla \mathbf{u} + (\nabla \mathbf{u})^T] \) is the stress tensor and \( \nu \) the kinematic viscosity.

In order to treat heat transfer problems the system (3) is equipped with a thermal diffusion
equation and thermal diffusivity \( \alpha \):

\[ \frac{\partial T}{\partial t} + (\mathbf{u} \cdot \nabla) T = \nabla (\alpha \cdot \nabla T). \quad (4) \]

The primal flow field can be described by the state vector \( \mathbf{s} = (\mathbf{u}, p, T)^T \), which is a solution
of the system (3,4). Since we focus on steady state solutions, we omit the time-derivatives and
rewrite the system in residual form:

\[ \begin{bmatrix}
\mathbf{r}_1 \\
\mathbf{r}_2 \\
\mathbf{r}_3 \\
\mathbf{r}_4 \\
\mathbf{r}_5
\end{bmatrix} = \begin{bmatrix}
(\mathbf{u} \cdot \nabla) \mathbf{u} + \nabla p - \nabla \cdot [2\nu \mathbf{D(u)}] \\
\mathbf{u} \cdot \nabla T - \nabla (\alpha \cdot \nabla T)
\end{bmatrix} = \mathbf{0} \quad (5) \]

4 THE ADJOINT FORMULATION

In order to derive the adjoint system and the desired sensitivities we have to formulate an
augmented cost function which obeys the governing equations as a constraint. This leads to a
formulation of the adjoint system and adjoint boundary conditions. The necessary procedure
will be demonstrated in the following.

4.1 The augmented cost function

The general approach of deriving the adjoint sensitivity analysis starts from an augmented
objective function \( L \), which is based of the cost function \( I \) augmented by the residual form of
the state equation \( \mathbf{r}(\mathbf{s}, \beta) = \mathbf{0} \), and the adjoint state variables \( \mathbf{\hat{s}} \) acting as Lagrange multipliers.
This approach is meaningful since the fulfilment of the governing equations acts as a constraint
on the optimisation problem.

\[ L(\mathbf{s}, \beta) = I(\mathbf{s}, \beta) + \int_{\Omega} \mathbf{\hat{s}}^T \mathbf{r} \, d\Omega. \quad (6) \]

Since (6) depends on the solution of the continuous flow field \( \mathbf{r} \) and the current design expressed
by the vector of design variables \( \beta \) the total variation of \( L \) due to a change in \( \Omega \) is

\[ \delta L = \left( \frac{\partial I}{\partial \mathbf{s}} + \int_{\Omega} \mathbf{\hat{s}}^T \frac{\partial \mathbf{r}}{\partial \mathbf{s}} \, d\Omega \right) \delta \mathbf{s} + \left( \frac{\partial I}{\partial \beta} + \int_{\Omega} \mathbf{\hat{s}}^T \frac{\partial \mathbf{r}}{\partial \beta} \, d\Omega \right) \delta \beta. \quad (7) \]

Choosing \( \mathbf{\hat{s}} \) such that the first right hand side term in (7) vanishes identically, i.e. for all
admissible states \( \mathbf{s} \), we write:
\[
\frac{\partial I}{\partial s} \delta s + \int_{\Omega} \hat{s}^T \frac{\partial r}{\partial s} d\Omega \delta s. = 0.
\] (8)

This choice of the adjoint state vector \( \hat{s} \) motivates the alternative viewpoint of the adjoint variables as Lagrangian multipliers [4]. Consequently, the sensitivity of (6) reduces to
\[
\delta \beta L = \frac{\partial I}{\partial \beta} \delta \beta + \int_{\Omega} \hat{s}^T \frac{\partial r}{\partial \beta} \delta \beta \ d\Omega = \delta \beta r.
\] (9)

### 4.2 Sensitivity of the cost function

The sensitivity of the cost function with respect to the design parameters (9) can be reformulated in order to shift the variation of the state vector from \( \beta \) to \( s \). By using the fact that the variation of \( r \) vanishes for any admissible variation of \( s \), we deduce
\[
0 = \delta r = \delta \beta r + \delta_s r,
\] (10)

and thus
\[
\delta \beta r = -\delta_s r.
\] (11)

Substituting (11) into (9) yields
\[
\delta \beta L = \delta \beta I - \int_{\Omega} \hat{s}^T \delta_s r \ d\Omega.
\] (12)

Now we are able to calculate (9) by the variation due to the design parameters and an inner product between the variation of the governing equation with respect to the design variables \( \beta \) and the adjoint state vector \( \hat{s} \). The latter is just the solution of the adjoint system, i.e. the solution of our adjoint flow equation.

### 4.3 The adjoint equation system

Starting point of the adjoint approach is variation of the Lagrange function (8). This variation has to be identically zero, i.e.
\[
\delta \beta L = \delta_s L + \int_{\Omega} \hat{s}^T \delta_s r \ d\Omega \equiv 0.
\] (13)

The vector \( \hat{s} \) can be interpreted from two different viewpoints: as vector of the Lagrange multipliers, or as adjoint state vector with adjoint velocity, adjoint pressure and adjoint temperature, i.e. \( \hat{s} = (\hat{u}, \hat{p}, \hat{T})^T \).

After several transformations and the demand that the derived equations have to be fulfilled for all variations of the primal state one derives the corresponding inhomogeneous continuous adjoint system (see [13] for details):
\[
\mathbf{D}(\hat{u}) \mathbf{u} + \nabla \cdot (2\nu \mathbf{D}(\hat{u})) - \nabla \hat{p} + T \nabla \hat{T} = \frac{\partial I_\Omega}{\partial \mathbf{u}},
\]
\[
\nabla \cdot \hat{u} = \frac{\partial I_\Omega}{\partial \hat{p}},
\]
\[
\mathbf{u} \cdot \nabla \hat{T} + \nabla \cdot (\alpha \nabla \hat{T}) = \frac{\partial I_\Omega}{\partial \hat{T}}.
\]

(14)

In the case one assumes only surface contributions, i.e. one focusses on shape rather than volume or topology optimisation, one has to deal with the homogeneous adjoint system:

\[
\mathbf{D}(\hat{u}) \mathbf{u} + \nabla \cdot (2\nu \mathbf{D}(\hat{u})) - \nabla \hat{p} + T \nabla \hat{T} = 0,
\]
\[
\nabla \cdot \hat{u} = 0,
\]
\[
\mathbf{u} \cdot \nabla \hat{T} + \nabla \cdot (\alpha \nabla \hat{T}) = 0.
\]

(15)

### 4.4 Adjoint boundary conditions

In order to close the system (15) we have to fulfil appropriate boundary conditions.

Here one distincts between conditions which have to be fulfilled at the different types of boundaries, i.e. inlet, wall and outlet, in order to incorporate the different situation at each boundary type (see [7]).

To this use we split the adjoint velocity vector into tangential and normal parts, i.e.

\[
\hat{u} = \hat{u}_t + \hat{u}_n = u_t t + u_n n \quad \text{with} \quad t \perp n.
\]

(16)

With these assumptions and appropriate primal boundary conditions, we derive boundary conditions for the adjoint variables at the inlet, wall and outlet:

**Inlet**

\[
\hat{u}_t = 0, \quad \hat{u}_n = \frac{\partial I_\Gamma}{\partial \hat{p}}, \quad \frac{\partial \hat{p}}{\partial n} = 0, \quad \hat{T} = 0.
\]

(17)

At the inlet, we assume Dirichlet conditions for primal velocity and primal temperature, and Neumann conditions for the primal pressure.

**Wall**

\[
\hat{u}_t = 0, \quad \hat{u}_n = -\frac{\partial I_\Gamma}{\partial \hat{p}}, \quad \frac{\partial \hat{p}}{\partial n} = 0, \quad \frac{\partial \hat{T}}{\partial n} = -\frac{1}{\alpha} \frac{I_\Gamma}{\partial \hat{T}}.
\]

(18)

At walls, we assume no-slip conditions for the primal velocity, Dirichlet conditions for the primal temperature, and Neumann conditions for the primal pressure.

**Outlet**

\[
u u_n \hat{u}_t + \nu (n \cdot \nabla) \hat{u}_t = \frac{\partial I_\Gamma}{\partial u_t},
\]
\[
\mathbf{u} \cdot \mathbf{u} + u_n u_n + \nu (n \cdot \nabla) \hat{u}_n + T \hat{T} + \frac{\partial I_\Gamma}{\partial u_n} = \hat{p}
\]
\[
\frac{u_n \hat{T}}{\partial n} + \alpha \frac{\partial \hat{T}}{\partial n} = \frac{\partial I_\Gamma}{\partial \hat{T}}
\]

(19)

The boundary conditions are generally in the form that they contain surface contribution \( I_\Gamma \). Choosing a concrete cost function leads to specific adjoint boundary conditions.
5 OPTIMIZATION PROCEDURE

Obviously the optimization process depends on the overall design goal, i.e. minimizing or maximising the cost function. The link between optimization of the cost function and parameter variation is the total variation of the augmented cost function (6), i.e. the sum of the total variation with respect to the state vector $s$ and the design parameters $\beta$:

$$\delta L = \delta_s L + \delta_\beta L. \quad (20)$$

The variation with respect to the state vector $s$ directly leads to the solution of the adjoint system and was demonstrated in the foregoing section. What remains is the variation of the Lagrangian $L$ with respect to the normal displacement of the boundary, which was

$$\delta_\beta L = \delta_\beta I + \partial_n s \cdot \partial \beta \int_\Omega s^T r \Omega. \quad (21)$$

These are the sensitivities of the system with respect to the design parameter $\beta$. They keep the essential information how to deform the geometry in order to improve the cost function.

5.1 Sensitivities

In order to derive the sensitivity information one has to evaluate the variation of the state vector $s$ with respect to the surface variation. Following the approach in [5] we linearised around a surface node $x_n$ with $s(x_n + \beta_i) = s(n) + \partial \beta_i + O(\beta_i^2)$ and approximate the variation as

$$\delta_\beta s = \delta_\beta s \cdot \delta \beta \approx \partial_n s \cdot \delta \beta \quad (22)$$

This yields locally

$$\delta_\beta L = \delta_\beta I + \partial_n s \cdot \partial_n I \Gamma \quad (23)$$

Following the derivation in [7], i.e. considering no-slip condition at walls and the adjoint boundary conditions, the local surface sensitivity of a normal displacement of the surface is

$$\frac{\partial L}{\partial \beta_i} \approx -\nu \frac{\partial u_i}{\partial n_i} \frac{\partial u_i}{\partial n_i} - \alpha \frac{\partial T}{\partial n_i} \frac{\partial T}{\partial n_i} =: \sigma_i \quad (24)$$

with sensitivity vector $\sigma = (\sigma_1, \ldots, \sigma_N)^T$, corresponding surface normal vector $n_i$ and $N$ the number of surface nodes.

5.2 Mesh deformation

The sensitivities represent the information how to deform the geometry, especially in which direction: inward or outward movement of the surface node. In general, the resulting sensitivity vector field will be highly distorted. In order to derive a smooth deformation vector field and thus a continuous geometry update one has to smooth the sensitivity field $\sigma$. This is usually carried out by solving a Laplace equation for $\sigma$ with a suitable diffusivity $\gamma$:

$$\nabla \cdot (\gamma \nabla \sigma) = 0 \quad (25)$$

which results in a smoothed sensitivity field $\tilde{\sigma}$, which is used to perform the mesh update. The mesh update is carried out with a standard mesh motion approach in OpenFOAM.
5.3 The general design cycle

The overall goal of the design process is to derive a method of a successive improvement of the cost function based on the derived sensitivity information from the solution of the primal and the adjoint system. This procedure is an iterative process, meaning we have to apply the adjoint-based sensitivity analysis to the updated geometry in a repeated manner until it is converged or a prescribed design goal is reached. The resulting design loop can be formulated via the following steps:

For $i:=1,N$

- Step 1: Solve primal equation system (5).
- Step 2: Solve the adjoint equation system (15).
- Step 3: Compute the sensitivity information (24).
- Step 4: Update the geometry based on Step 3.
- Step 5: Evaluate the cost function $I_i$.
- Step 6: Proceed if $|I_i - I_{i-1}| > \varepsilon$

The ingredients of this optimization cycle, i.e. primal and adjoint solver, sensitivity computation, cost function evaluation and mesh deformation, are implemented in and use tools from the open-source CFD toolbox OpenFOAM [14].

6 APPLICATION

As a proof-of-concept we apply the approach to a concave dimpled plate [15]. Compared to ribs and fins, dimpled geometries show the best thermal-hydraulic performance defined as the ratio between the heat exchange and the pressure loss (see [16] for details). Our design goal is to maximise the wall heat flux on the lower boundary with the dimple.

6.1 The domain

We start from a rectangular domain with length $x = 0.276$ m, width $y = 0.08$ m, and height $z = 0.03$ m. The lower and upper faces are wall boundaries, the lateral ones are handled as cyclic boundaries. The lower boundary is equipped with a dimple with diameter $d = 0.048$ m and height $h = 0.012$ m. The overall simulation domain and the lower wall with dimple are represented in Figure 1.

Initial conditions for the simulations at the inlet are: velocity $U = 5$ m/s and temperature $T = 330$ K. At wall boundaries we have: velocity $U = 0$ m/s and temperature $T = 293$ K.

6.2 Cost function

The design goal is to maximise the heat flux at the lower boundary. Consequently, the cost function is chosen as the integral of the heat flux through the bottom face i.e. the normal derivation of the temperature field on this patch:

$$ I = \int_{\text{wall}} \frac{\partial T}{\partial n} \, d\Gamma $$

(26)
6.3 Results

In the following we present and discuss the results of the application of the optimization cycle from section 5.3 to the above described geometry in order to improve the cost function (26).

Figure 2 represents the improvement of the wall heat flux on the lower boundary due to the design cycles iterations. It is worth to note that the pressure loss of the optimized geometry is merely increased by approx. 3%, while the wall heat flux is improved by approx. 15%.

A comparison between the shape of the initial dimple geometry and the optimized one is depicted in Figure 3. It can be clearly seen that the algorithm flattens the upstream edge of the dimple geometry while the downstream edge is slightly raised.

Flow fields for velocity and temperature of simulations of the initial and the optimized dimple geometry are depicted in Figure 4 and Figure 5, respectively. One clearly sees how velocity and temperature propagate into the dimpled domain, which results in an improved heat-flux at the lower boundary.
Thorsten Grahs and Johann Turnow

7 SUMMARY AND OUTLOOK

We derived a continuous adjoint formulation of the steady-state, incompressible Navier-Stokes equations augmented with a diffusion equation for the temperature. The derived general adjoint system was adapted to specific maximisation of the heat flux at walls. The adjoint solver and adjoint boundary conditions were implemented into the CFD toolbox OpenFOAM in order to derive an optimization process involving mesh deformation based on the adjoint sensitivity.
analysis derived from the primal and the adjoint solution of the system.

As a proof-of-concept this optimization approach was applied to a dimpled channel geometry. The results presented here are quite promising, since we were able to increase the wall heat flux on the lower boundary by approx. 15% while the pressure drop just increased slightly by approx. 3%. Nevertheless, further improvement and validation of the approach is necessary.

Future work will focus on the mesh deformation algorithm, e.g. with a radial-basis function approach, as well as the incorporation of additional cost function and a combination of these cost functions. Especially the combination of maximising heat-transfer and minimising pressure loss will be the next research topic in order to further improve heat exchangers for ducted flows.

REFERENCES


MULTI-OBJECTIVE SURROGATE BASED HULL-FORM OPTIMIZATION USING HIGH-FIDELITY RANS COMPUTATIONS

Thomas P. Scholcz*, Christian H.J. Veldhuis
Maritime Research Institute Netherlands (MARIN)
Haagsteeg 2, 6708 PM Wageningen, The Netherlands
e-mail: t.p.scholcz@marin.nl, web page: http://www.marin.nl

Key words: Optimization, Hull-form, Multi-objective, Multi-level, Surrogate, RANS

Abstract. RANS-based optimization procedures for ship design become increasingly complex and require the development of more efficient optimization techniques. The four phases of the design procedure are: shape parameterization, global sensitivity analysis, multi-objective optimization and design review. The dimensions of the design space can be mitigated by a smart choice for the shape parameterization and by screening and ranking the design variables in the global sensitivity phase. Subsequently, Surrogate Based Global Optimization (SBGO) is used to reduce the cost of the multi-objective optimization phase. For a practical application it is shown that the computational time reduces from two weeks to only a day when using SBGO instead of applying a Multi-Objective Genetic Algorithm (MOGA) directly to the solver. The design review phase is then used to verify and further develop the optimal design. Here, we focus on automatic ship design techniques which comprises the first three steps of the design procedure. Accelerating the ship design process is subject of ongoing research at the Maritime Research Institute Netherlands, making it useful for practical applications with turnaround times of only a few weeks.

1 INTRODUCTION

Automatic RANS-based optimization procedures are becoming increasingly important in practical ship design. However, due to its complexity this type of optimization is very computationally demanding. In order to reduce the computational burden Surrogate Based Optimization (SBO) can be used. A number of studies demonstrated the potential of surrogate acceleration techniques. In [1] surrogates are used to obtain approximate Pareto fronts of a chemical tanker. A number of surrogate techniques were studied including Kriging, universal Kriging and polynomial regression. It was found that the ship design process could be accelerated leading to more efficient ships. In [2] a procedure is discussed that aims to obtain minimum required power and best wake field quality using viscous flow computations. Design of Experiments and generic hull shape variations were used to speed up the optimization process. In [3] the effect of numerous hull form variations and condition variations were studied. Surrogate models were used for each
water depth condition in order to make the final design trade-off. This contribution aims to make the step to the actual automatic optimization of ships. The previous studies mostly focussed on design space exploration with a clever usage of Design of Experiments. Coupling this to an optimizer can improve the final design and ease the optimization process. To show this an overview of how SBO can be used in practical optimization projects is given. Starting with a base design, four important phases of the design procedure are identified: Shape parameterization: e.g. using generic hull shapes or a set of predefined hull variants (Section 2). Global Sensitivity Analysis: screening of the design space and ranking of the design parameters (Section 3). Multi-Objective Optimization and surrogate acceleration techniques (Section 4). Design review: verify the optimal designs and choose/modify the design if necessary. Although the last step is crucial in the design procedure it is not discussed in this contribution. Here, we will focus on computerized ship design techniques which comprises the first three steps of the design procedure.

2 SHAPE PARAMETERIZATION

For the design space definition we used a B-spline-Merge method [8] for the parametric deformations of the geometry. This method is implemented in the CAD-tool Rhinoceros and is referred to as Rhino-Merge. Rhino-Merge interpolates between some basis hull forms. These basis hull forms span the design space. Figure 1 illustrates how Rhino-Merge can make combinations of the basis hull forms by making linear combinations. Here an example of two basis hull forms is shown of which the average is taken as final shape.

From here on a designer can choose to generate the basis hull forms manually, based on experience or initial CFD calculations. There is also an option to generate basis hull forms in a more generic way. Figure 2 illustrates the set up of the generic hull shapes. The generic hulls shapes are set up in two ways: In order to shift displacement the more widely used Lackenby shift method is used (see [10]), simply by moving the sections with a predefined function. In order to change the local shape of the hull the sections are modified. These modifications can be done in several ways aiming at independent (hence an orthogonal design space) and realistic shape variations; currently we choose Chebyshev mode variations. Note, these modes can be convenient to use for single screw vessels. Other ship types (e.g. prame type ferries, yachts) need different generic functions to result in proper shape variations.
A disadvantage of using generic hull shapes is that the number of basis hull shapes can quickly become too large to handle in combination with an optimizer. With manually designed basis hull shapes this is less of a problem. But, still the curse of dimensionality can set in quite fast when multiple design directions should be looked at. Therefore a sensitivity study is done to detect the most promising generic basis hull shapes.

3 GLOBAL SENSITIVITY ANALYSIS

Global sensitivity analysis is a useful approach to learn about a design problem before an optimization procedure is initiated. One distinguishes between local and global sensitivity analysis. In a local sensitivity study one aims to obtain the partial derivative at a specific point in the design space. This derivative can be computed using an adjoint method or approximated using finite differences. In a global sensitivity study one aims to obtain general trend data over a whole range in the design space. This data is obtained via sampling and regression.

For this study a tanker is taken from the 7th-Framework EU project STREAMLINE. The ships speed is 14 knots, Lpp=94m, B=15.4m, the design draft is 6m and the block coefficient is 0.786. The Froude number is 0.237 and the Reynolds number is 6x10^8. More on the optimization of this ship can be found in [2]. The 9-dimensional design space was created by means of generic basis hull forms. Two objectives were chosen: the ship resistance (resistance coefficient) and the wake quality (Wake Object Function) calculated with the structured RANS code PARNASSOS ([13, 14]). In this sense a balance (compromise) can be made between resistance and comfort level. Note, a better objective instead of resistance would be the power. However, because the goal was to test several optimization techniques it was decided to minimize computational effort.

3.1 Partial correlations

An initial Latin Hyper Cube Design of Experiment consisting of 90 PARNASSOS evaluations (10 per dimension) is used to scan the design space. The calculations were performed in parallel and took about 1 day on the MARIN cluster. Dakota ([5]) is used to generate the Latin
Hypercube Design and to automatically obtain the partial rank correlations, see Figure 3. This data is obtained by calculating Spearman’s rank correlation coefficients on the Design of Experiment. When an objective is increasing with a design variable the correlation is positive. When the objective and the design variable are related by a monotonic function, Spearman’s coefficient becomes equal to one.

Looking at Figure 3 we can now distinguish between design variables that result in conflicting/non-conflicting objectives or have only a limited effect on the objectives. For example, design variable nine shows a strong positive correlation with both resistance coefficient and Wake Object Function. This is an indication that the variable can be ignored in the optimization study which leads to dimensionality reduction.

3.2 Scatter plots

Scatter plots help to interpret the correlations from Figure 3 by visualizing the data along with the trends. Figure 4 shows a scatter plot of the two objectives: Resistance coefficient (Obj1) and Wake Object Function (Obj2). Note that the slopes of the trends correspond to the correlations in Figure 3.
If design parameters have a strong interaction, scatter plots and partial correlations can be deceptive. In this case interaction detection methods are required such as variance based decomposition to reduce the dimensions of the optimization problem, see [9]. However, the interactions are usually small when the mode shapes are geometrically orthogonal. This is the case for the Chebyshev mode variations used in this contribution.

4 MULTI-OBJECTIVE OPTIMIZATION AND SURROGATE ACCELERATION TECHNIQUES

Multi-objective optimization arises naturally in ship design problems since multiple objectives need to be optimized that may or may not conflict with each other. A classical example is the optimization of the wave resistance defined at several conditions that approximate a ship’s operational profile, see [11].

Figure 4: Scatter plots and partial linear trends. The slopes correspond to the correlations in Figure 3.

Figure 5: Surrogate acceleration method from [4].

Figure 6: Dakota interface, see [5].
The increasing computational complexity of practical optimization problems results in high dimensional design spaces that cannot always be reduced by dimension reduction techniques. When the dimensions are not too high, surrogate based acceleration techniques can be used to mitigate the computational effort of the optimization, see for example the acceleration scheme shown in Figure 5. This algorithm is implemented in Sandia’s optimization toolkit Dakota ([5]) and can be used once the interface between the in-house simulations code ReFRESCO or PARNASSOS is established, see Figure 6. ReFRESCO is an unstructured state-of-the-art viscous-flow RANS code while PARNASSOS is a structured steady viscous-flow RANS code. The choice of the solver depends on the application and budget of the design project. If the optimization is a pure trade-off of (conflicting) objectives that do not depend on other optimizations it is named a single level optimization. This is the topic of Section 4.1. For some practical problems, e.g. hull-propeller optimization, two or more levels of optimization exist. These so called multi-level optimization problems often require an extreme number of expensive code simulations. Reducing the computational effort of such problems with surrogate acceleration is a challenge. This is the topic of Section 4.2.

4.1 Single-level optimization

In this section we study the single-level optimization problem defined in Section 3. The objective functions that need to be minimised are the resistance coefficient and the Wake Object Function obtained from the ”streamline” tanker sailing at a speed of 14 knots (Fn = 0.237). First, a direct Multi-Objective Genetic Algorithm (MOGA) is used on the simulation code PARNASSOS in order to obtain the true Pareto front. Second, two strategies are used to obtain approximate Pareto fronts: surrogate based optimization on the initial surrogate (without adding new designs) and surrogate optimization on updated surrogates (adding new designs).

4.1.1 Direct Multi-Objective Genetic Algorithm

The results of the MOGA are shown in figure 7. The optimization progresses towards the onset of the true Pareto front. However, after 2 weeks (and over 300 PARNASSOS evaluations) the process stopped due to time constraints. Note, these evaluations cannot fully be done in parallel because all evaluations of a MOGA population should finish before evaluations of the next population can start.
Figure 7: Pareto plot for resistance versus wake quality (Wake Object Function, WOF) for a direct MOGA.

4.1.2 Surrogate based optimization on the initial surrogate

As a next step surrogates were obtained from the initial Latin Hyper Cube design used in Section 3.1. The surrogates were constructed using a quadratic polynomial fit and universal Kriging with quadratic trend. Subsequently, MOGA was used on the quadratic polynomial fit to obtain an approximate Pareto front. Figure 8 shows the populations of this simulation in the objective space. Like for the direct MOGA the populations steadily progress towards a Pareto front.

Figure 8: Pareto plot for resistance versus wake quality (Wake Object Function, WOF) for a MOGA on an initial response surface.
Figure 9 shows the comparison between the Pareto front from the direct MOGA approach and that from the approximate Pareto fronts using the quadratic polynomial and universal Kriging. Clearly the way the surrogate is built influences the approximate Pareto front. Still, both approximations are close to the true MOGA front while they were computed in only a day as apposed to the two weeks required by the direct MOGA approach.

![Figure 9](image)

**Figure 9**: Pareto plot for resistance versus wake quality (Wake Object Function, WOF) for a direct MOGA, a MOGA on a quadratic response surface, and a MOGA on a universal kriging response surface.

The question arises whether the surrogate can be updated with new designs in order to improve its quality. This leads to Surrogate Based Global Optimization (SBGO) on updated surrogates as discussed next.

### 4.1.3 Surrogate Based Global Optimization on updated surrogates

A more elaborate SBGO takes the resulting Pareto front, determines the new designs to be evaluated, and adds those results to the dataset in order to update the surrogate. On the new surrogate a MOGA is done to determine an updated Pareto front. This process is visualized in Figure 10 showing the initial Design of Experiment, the front obtained on the initial surrogate, true values of the new designs that are selected from the initial surrogate front, the front obtained on the updated surrogate and finally the true front obtained with direct MOGA. All results are obtained using the quadratic polynomial fit.
Figure 10: Pareto plot for resistance versus wake quality (Wake Object Function, WOF) using SBGO on updates surrogates.

Note that the true values of the new designs that are selected from the initial surrogate are already quite close to the true front, as expected. The resistance coefficients of the new designs are a bit smaller than the true values and this leads to an updated surrogate front with slightly lower resistance coefficients. The new front is more compact (all resistance coefficients smaller than $2.55 \cdot 10^{-3}$) and more aligned with the true front at the cost of only a few additional calculations. Since the true values of the new designs are already slightly better than the values of the true front obtained with MOGA it is suspected that the latter was not entirely converged when it reached its maximum number of evaluations.

A practical issue with these type of optimizations is the importance of failure capturing and its effect on simulation robustness. The failure capture techniques available in Dakota are abort, retry, recover and continuation. The recover method returns a dummy value to Dakota in case of failure and the continuation method searches for nearby parameters that do not fail. Since surrogates can be sensitive to outliers, these methods are sometimes not satisfactory and manual intervention is required to obtain the desired results. Knowledge of the simulation robustness is therefore critical for the success of (automatic) surrogate based optimization methods.
4.2 Bilevel optimization

Bilevel optimization is a branch of optimization, which contains a nested inner optimization problem within the constraints of an outer optimization. These inner and outer optimization problems are also called the upper level and lower level respectively. In naval ship design such problems arise naturally from sub-systems that interact with and influence each other, see [6]. Due to the complexity of these type of optimization problems surrogates are often used to mitigate the computational effort. Depending on the type of interaction between the sub-systems one can replace either the lower level objectives, upper level objectives or all objectives by surrogates, see [7]. Here we consider the optimization of the shape of a twin screw open shaft vessel and propeller, the upper and lower optimization level respectively, see Figure 11.

![Figure 11: From right to left (downstream): Variations of the hull shape, shaft orientation, and propeller diameter. Nominal wakefields calculated for each hull form with the RANS solver ReFRESCO. Lower level: propeller optimization using PropArt with the nominal wakefields as input.](image)

The overall goal is to find the orientation and shape of the hull/propeller combination that corresponds to minimum required power for propulsion and maximum comfort within the operational profile. Each optimization problem can be driven by its own dedicated software. We use the optimization toolkit Dakota ([5]) to drive the upper level optimization problem on the CFD code ReFRESCO and an in-house optimization tool called PROPART ([12]) for the lower level optimization problem on the simulation code PROCAL ([16]). The nominal wakefield that results from an upper level evaluation acts as an input to the lower level propeller optimization. Finally, two-way coupling is used between ReFRESCO and PROCAL (with the optimized propeller) in order to take into account interaction effects. Surrogate acceleration techniques as discussed in Section 4.1.2 and 4.1.3 are used to reduce the upper level evaluations of the optimization. This study is part of ongoing research at MARIN. New results will be published in future publications.

---

1ReFRESCO (www.refresco.org) is a community based open-usage CFD code for the Maritime World. It solves multiphase (unsteady) incompressible viscous flows using the Navier-Stokes equations, complemented with turbulence models, cavitation models and volume-fraction transport equations for different phases ([15]).
5 CONCLUSIONS

Four important phases of a ship design procedure are identified: shape parameterization, global sensitivity analysis, multi-objective optimization and design review. The first three phases comprise automatic ship design methods. Acceleration of these techniques will increase their practical applicability.

The shape parameterization can be defined by linear combination of predefined hull shapes or by using generic hull shapes. It is found that the number of required generic hull shapes quickly becomes too large which motivates the use of global sensitivity analysis and dimension reduction techniques.

Partial correlations and scatter plots can be obtained by sampling and regression. When interactions are not too strong this data can be used to reduce the dimensions of the design-space. This is usually the case when geometrically orthogonal hull shapes are used in the shape parameterization.

The complexity of present RANS-based multi-objective optimization problems calls for optimization techniques that reduce the required computational effort. Surrogate Based Global Optimization (SBGO) is a promising approach to mitigate the computational burden that results from high dimensional design spaces and/or multi-level (nested) optimization problems that arise naturally in naval ship design. For a practical application we showed that SBGO reduces the required computational time from two weeks to only a day when compared with a direct Multi-Objective Genetic Algorithm. It is however crucial to have knowledge about the robustness of the simulation code and to properly capture simulation failures during the SBGO process.

This knowledge and experience is currently acquired in several ongoing projects at the Maritime Research Institute Netherlands.

6 Acknowledgement

This research was funded from the TKI-allowance of the Dutch Ministry of Economic Affairs. The support is gratefully acknowledged.

REFERENCES


PROPELLER NOZZLES DESIGN USING VISCOUS CODES AND OPTIMIZATION ALGORITHMS

STEFANO GAGGERO*, DIEGO VILLA*, GIORGIO TANI* AND MICHELE VIVIANI*

* Department of Electrical, Electronic, Telecommunications Engineering and Naval Architecture
University of Genoa
16145 Genoa, Italy
www.unige.it

Key words: Ducted Propellers, Accelerating Nozzles Design, Decelerating Nozzles Design, Optimization, OpenFOAM.

Abstract. Marine propellers design requirements are always more pressing and the application of unusual propulsive configurations, like ducted propellers with decelerating nozzles, may represent a valuable alternative to fulfil stringent design constraints. If accelerating duct configurations were realized mainly to increase the propeller efficiency in highly-loaded conditions, decelerating nozzles sustains the postponing of the cavitating phenomena that reflects into reduction of vibrations and radiated noise. The design of decelerating nozzle, unfortunately, is still challenging. No extensive systematic series are available and the design relies on few measurements. On the other hand, viscous flow solvers appear as reliable and accurate tools for the prediction of complex flow fields. Hence, in the present paper the opportunity to use CFD as a part of a design procedure based on optimization, by combining a parametric description of the geometry, the OpenFOAM solver and a genetic type algorithm in the ModeFrontier environment, is investigated. Design improvements for both accelerating and decelerating ducts are measured by comparing the performance of the optimized geometries with those of conventional shapes available in literature.

1 INTRODUCTION

Marine propellers design requirements are, nowadays, always more pressing. Not only maximum efficiency, but also comfort and environmental demands and regulations have to be satisfied. The application of unusual propulsive configurations may represent a valuable alternative to fulfil these constraints. Contra-rotating and tip loaded blades (CLT and Kappel like geometries) were used to improve propulsive efficiency, together with Energy Saving Devices, like pre- and post- swirl stators, PBCF or Mewis Ducts. Combinations of these devices mounted on PODs (i.e. through the EU project TRIPOD, [1]) were exploited to maximize the effects and to provide flexible propulsive systems able also to reduce side effects like induced pressure pulses and vibrations by operating the propellers in more uniform inflow, with minimum optimal diameters and far from the hull. Ducted propellers represent an additional, valid, answer to current design requirements.

In the case of vessels for which requirement of high thrust are critical in the low speed
operation range, or when the screw is limited in diameter, the use of accelerating nozzles is widely documented in literature: with accelerating nozzles, the duct increases the flow rate through the propeller, which consequently operates at a more favourable loading. The nozzle by itself produces a certain positive thrust.

This conceptual idea dates back to Stipa [2] and Kort [2] but probably the works by Van Manen [4], Van Manen and Superina [5] and Van Manen and Oosterveld [6] represent some of the most accurate reviews of ducted propellers performance and design guidelines. By combining even simple theoretical considerations and systematic measurements, they provided a rather wide overview of the influence of some geometrical parameters of nozzles on efficiency, correlating non-dimensional parameters and performance in case of both highly- and lightly-loaded propellers. They identified the nozzle profile number 19A as the optimal compromise, having performance in towing and pushing conditions not appreciably inferior to those of considerably longer nozzles. This shape, fifty years later, is still the default choice in the case of accelerating ducted propellers applied to a variety of boats and vessels (tugboats, towboats, and supply vessels) which require increased propulsive efficiency especially near the bollard pull condition.

In the same works, also the potentialities of flow decelerating type of nozzle were pointed out. By reducing the flow rate to the impeller, a local increase of the static pressure is achieved, which could be effective in retardation of propeller cavitation phenomena at a cost, unfortunately, of efficiency reduction and an additional drag represented by the negative thrust delivered by the duct itself. An improvement of the cavitation characteristics of the propeller can be obtained, thus, only if the gain in static pressure exceeds the unfavourable effect of the increased screw loading, necessary to compensate the duct drag. In addition, the risk of duct cavitation (at leading edge or midchord, depending on the propeller loading and the camber of the nozzle) represents another issue to account for in evaluating the usefulness of the decelerating configuration.

The design of decelerating nozzles, which consequently seem to better comply with the constraints circa radiated noise and vibrations strictly related to cavitation suppression, is however still challenging and poorly addressed in literature. Despite the wide employment of this propeller configuration (i.e. fast carriers and navy vessels for which side effects reduction is significantly more important than propulsive efficiency), there are several design and analysis issues still open. No systematic series, widely diffused in the case of accelerating configurations, are available and the design still relies on few measurements and data. From a theoretical/numerical point of view, the availability of literature data is even scarce. Apart classical momentum theories [6] applied to decelerating duct geometries, only few applications are available. Abdel Maksoud et al. [7] developed a design and analysis method for multi-component propulsors. Gaggero et al. [8] developed a design approach based on optimization through Boundary Elements Methods. Results, confirmed by dedicated RANSE calculations and model scale measurements, concerned however only the well-established problem of propeller blades design, using given nozzle shapes. The inclusion of the duct shape in the optimization process, certainly possible, would have excessively stressed, over the inherent limitations of the potential approach, the applicability of BEM to predict nozzle forces, significantly influenced by viscous effects and flow separation with the risk of obtaining shapes conditioned by the low fidelity level of the computational tool. Recently, yet in the framework of potential flow based theories, some analyses of decelerating duct shapes
with momentum theory and semi-analytical actuator disk models to account for radial distributions of forces have been proposed [9]. All these calculations, however, neglect the influence of viscosity which instead, also in the simpler case of accelerating nozzles [8,10,11] significantly influence duct and propeller forces by interacting with the flow at the duct/propeller tip gap giving rise to typical tip leakage vortexes. 

As the consequence of the need of a reliable and efficient design tool for accelerating and decelerating propeller nozzles, in the present work optimization is exploited to investigate the possible enhancement of nozzles performance provided by the application of high-fidelity numerical tools embedded into an automatic process. Thousands of different geometries, defined by a parametric description, need to be tested and verified against appropriate design criteria (specifically derived for both accelerating and decelerating configurations) to define an optimal geometry that, due to the multi-objective nature of design, is of course a balance of contrasting purposes. Only thanks to the favourable ratio between accuracy and computational efficiency of simplified representations of the influence of the propeller on the nozzle performance, such the ones represented by actuator disks, the process is possible in a reasonable computational time compatible with the usual design routine. After a validation based on the measurements carried out at the cavitation tunnel of the University of Genoa, actuator disk models, already proposed with success for the analysis of accelerating ducts performance, will be therefore employed also for the analysis of decelerating ducts and, through optimization, for the improvement of the performance of reference nozzle geometries.

2 NUMERICAL METHODS

2.1 Flow solver and computational domain

Detailed analyses of duct performance cannot be achieved by simply using inviscid flow models. Flow separation, except by using heavily approximated formulations, is beyond potential flow calculations and blunt trailing edges, in the particular case of accelerating ducts, pose significant difficulties for the application of the Kutta condition. Viscous models are definitely needed and RANSE approximations with suitable turbulence models can adequately answer all the specific issues related to design procedures.

In this study, OpenFOAM [12] has been employed. OpenFOAM is a collection of libraries suited for the solution of partial differential equations using a Finite Volume approach and cell-centered collocated variables with face-based implementation to allow for arbitrary cell shapes. Pre-processing, solvers and post-processing tools are fully scriptable, which is of fundamental importance when a completely automatic process has to be setup. For this particular application, where steady flow with respect to a blade fixed reference frame is expected, the simpleFOAM solver with the SST $k-\omega$ turbulence model has been used: continuity and RANSE equations are solved in a segregated form, with a SIMPLE based pressure-velocity correction and under-relaxation to achieve steady state solutions. Second order accurate schemes in space have been preferred for momentum and turbulence equations.

The axisymmetric nature of the flow allows the use of “smaller” computational domains that, without the effective presence of the propeller blades substituted by distributions of the momentum sources of the actuator disk, could be reduced to a sector narrower than a blade passage. The simplest formulation available in OpenFOAM exploits “wedge type” boundary conditions for purely axisymmetric calculations. For 2 dimensional axisymmetric cases, the
geometry is specified as a wedge of small angle (<5°) and 1 cell thick running along the plane of symmetry, straddling one of the coordinate planes, as shown in Figure 1. With this arrangement, only momentum sources in the axial planes are allowed and all the flow features different from those on the meridian plane (i.e. slipstream rotation) are neglected. Slipstream flow has a certain importance [10], especially in correspondence to highly loaded functioning. In present case, the need of computationally efficient calculations and the “comparative” nature of the design by optimization encourage the use of the efficient setup consisting in the two-dimensional wedge formulation, thus neglecting the three-dimensional nature of the flow accounted, instead, with three-dimensional calculations and cyclic boundary conditions.

**Figure 1**: Computational domain with “wedge type” boundary conditions for the analysis of ducted propeller performance.

Also the propeller hub, in the shape of an infinitely long shaft, has been included in calculations. Its presence has a certain positive influence on the thrust delivered by the duct: momentum sources are distributed over a small area, inducing higher velocities that change the angle of attack of the nozzle and the resulting pressure distribution. Details of the hub shape would be also important. In present calculations, focused more on the definition of a procedure for nozzle shape design rather than on the exact characterization of the flow, the simplest adoption of an infinitely long shaft with a slip boundary condition was deemed sufficient to account at least for its principal influence. The final computational domain consists in an angular sector with the inlet placed 2 propeller diameters in front of the nozzle and the outlet 5 propeller diameter aft the nozzle. The external boundary lies on a cylindrical surface 4 propeller diameter from the propeller shaft.

### 2.2 Propeller model and grid arrangement

The influence of the propeller on the flow around nozzles has been modelled by using an actuator disk. The actuator disk can be seen, definitely, as an infinite bladed propeller. The load of the propeller is distributed over the entire area of the propeller, neglecting the real material nature of a finite number of blades. Even if more detailed representation of blade
forces are available [13], by using local momentum sources, their application for the characterization of flow in the nozzle design process seems redundant. Only the radial distribution of load can be considered important, influencing the risk of flow. In present calculations, the radial distributions of forces have been derived from Boundary Elements Method analyses and loaded into RANSE by using interpolating tables and polynomials through a dedicated library specifically developed to extend the computational functionalities of OpenFOAM.

In the light of dealing with flow separation, calculations have been carried out with a sort of “GAP” model, similar to that proposed in the case of BEM calculations and applied in the case of simplified viscous calculations [13] to easily account for the influence on loading of phenomena like tip leakage vortices. The proposed actuator disk is a finite thickness disk of width equal to the average thickness “cut out” by the propeller (Figure 2). Grids were arranged accordingly to the computational domain selected for the analyses by using the OpenFOAM `blockMesh` utility. A structured multi-block layout has been preferred to describe in details the most important features of the nozzles, to ease and to speed the automatic generation of the mesh possible thanks to the scriptable nature of the utility and necessary for optimization purposes. In the case of accelerating duct configuration, essentially an O-grid mesh has been setup to have better resolution at the blunt trailing edge. C-type grids were adopted, instead, in the case of decelerating geometries to manage the sharp trailing edge of the nozzle. The final arrangement consists in a one cell thick layer of 17600 structured hexahedral elements, setup around surfaces with appropriate grading to comply with wall treatment modelling through wall functions. An example is shown in Figure 2.

![Figure 2](image)

**Figure 2**: Actuator disk representation into the computational domain (a) and computational mesh (b) for the accelerating nozzle. Colours represent the strength of the momentum sources per unit volume.

### 2.3 Parametric description of nozzles

The parametric description of nozzles is rather simple, since nozzles are definitely hydrofoils that could be handled, analogously to what already done in the case of propellers [14] and wings [15], by using B-Spline curves for both camber and thickness distributions. In present calculations, however, a slightly different approach has been adopted. In order to avoid unfeasible shapes (i.e. maintain the inner duct surface on a cylindrical surface of a given radius) it was preferred to describe with B-Spline curves directly the inner and the outer contour of the nozzle to facilitate the application of the geometrical constraints [16].
6

Figure 3: Parametric description of nozzles shape. Accelerating duct case.

The B-Spline curves control points, as shown in Figure 3, are free to move with some “compatibility” constraints at leading and trailing edges. Furthermore, the control points that handle the central part of the inner surface have to be altered only axially to keep the nozzle shape rectilinear at propeller location. With these assumptions, the description of the accelerating duct consists in 11 free parameters while the decelerating nozzle has 7 degrees of freedom. In both cases, a scaling factor in the axial direction is added to change anisotropically the length/diameter ratio. Choices, also on the basis of previous experiences, are motivated by the need of a trade-offs between allowed modifications (the widest possible) and fairness of the geometry that is facilitated, in any case, by the use of B-Spline curves.

3 VALIDATION OF THE SIMPLIFIED MODEL

The validation of the computational tools has been carried out considering two ducted propellers for which measurements of forces and flow are available [8]. The first one is an accelerating duct propeller with a usual Nozzle 19A, designed for a take home propulsor. The second one is a decelerating duct propeller developed to postpone cavitation for applications in which reduction (or better, retardation) of noise is important, such as navy ones. In both cases, differently from usual Kaplan like geometries, propellers are unloaded at tip and have a small chord, which made the selection of the optimal nozzle shape problematic. By using a Boundary Elements Method, the radial distributions of axial force on the propeller blades were derived and momentum sources were assigned over the actuator disks coherently. Predicted forces exerted by the nozzles were compared to those measured during experiments. Results are shown in Figure 4. As usual thrust coefficients of propeller and nozzle ($C_T$ prop and $C_T$ Nozzle) made non-dimensional with respect to the inflow velocity $V$ and the area of the propeller disk) are compared in terms of thrust ratio $\tau$ in order to verify that the relative contribution of the duct to the total thrust increases with increasing propeller loading. Results, for both the nozzle geometries, are in good accordance with experimental data. Calculations well reproduce the experimental observations and from medium to heavy loading conditions, the agreement between calculations and measurements is good. The inclusion of the tangential forces as an additional momentum source distribution could have a favorable effect on the numerical calculations.
Stefano Gaggero, Diego Villa, Giorgio Tani and Michele Viviani

Figure 4: Comparison of measured and computed nozzle thrust. Decelerating duct. For confidentiality reasons, decelerating duct values are given non-dimensional with respect to a reference functioning condition.

As pointed out in Hoekstra (2006), the additional pressure drop in the swirling slipstream due to the modelling of tangential velocities increases the pressure difference between the inner and the outer duct surface, raising the duct forces to values closer to measurements [17]. Especially in the case of the accelerating duct geometry, the thrust of the nozzle is significantly overestimated in lightly loaded conditions.

Figure 5: Streamlines and inner pressure distribution at different loading conditions. Accelerating and decelerating ducts. For confidentiality reasons, the reference decelerating nozzle shape cannot be disclosed.

This difference may be explained by considering that the flow predicted by the actuator disk models is axisymmetric and neglects the tip leakage vortex, which can alter the streamlines path in circumferential direction and produce a negative interaction with the duct blunt trailing edge [13]. Streamline patterns and inner pressure fields for the two nozzle geometries are shown in Figure 5. The position of the leading edge stagnation point is evident, as well as how streamlines change with propeller loading. Streamlines approach Nozzle 19A almost along its camber line for a propeller thrust coefficient $C_T$ of about 4 while in lightly loaded conditions the risk of a separation bubble on the outer side is clear, completely in
agreement with the design of the nozzle, devoted to highly loaded functioning. For the decelerating nozzle shape, it is possible to appreciate the increase of the inner pressure close to the design point ($C_{T \text{ prop.}} / C_{T \text{ prop. ref.}} = 0.10$, $J = 103.4\%$ of $J_{\text{ref.}}$), together with streamlines approaching the nozzle at its ideal angle of attack.

3 OPTIMIZATION OF NOZZLES

The design of nozzles through optimization was achieved starting from two reference geometries available at the cavitation tunnel of the University of Genoa. In both cases, the design consists in the analysis of an initial population, selected by using the quasi-random Sobol sequencing, of 150 members, whose distribution in the design space supplies a sufficiently uniform description of the possible geometrical combinations. The optimization workflow, built in the ModeFrontier [18] environment, lets the initial population evolve for 15 generations that, in the light of preliminary calculations and the relative low number of free parameters, were considered sufficient to achieve convergence. The need to handle opposite objectives requires the use of multi-objective optimization algorithms that, in present calculations, are of genetic type. For an accelerating nozzle, which is usually designed to operate close to bollard pull, the maximization of the duct thrust at a given propeller loading conflicts with the need of reasonably high values of inner static pressure to contrast cavitation inception. For a decelerating duct propeller operating at relatively high advance coefficients, maximization of the static pressure at the minimum cost of increased duct drag turns into the design objective, to be achieved, in very critical functioning conditions, avoiding the risk of cavitation on the outer surface of the nozzle.

Avoiding local minima was considered important for the analysis of trends and of the influence of combinations of parameters. Genetic algorithms, with the inclusion of a certain randomness in the selection of the characters of any subsequent generations, allow more unrestrained analyses and a certain margin against local minima. A real design would require accounting for the interactions taking place between the duct and the propeller to exploit the maximum from both the designs. The use of actuator disk models (as proposed in this work) does not permit any type of interactions on propeller forces being these ones prescribed and independent of the force on the nozzle. Only by a blade redesign, it would be possible to have a sort of feedback and to design a propulsive system for a given total thrust. By assuming an unchanged propeller thrust (i.e. actuator disk intensity), instead, the analysis of the optimal geometries has to be restricted to those nozzle shapes that delivering the thrust of the reference geometry (i.e. not changing the functioning condition), improve performance in terms of inner static pressure. Any other case has to be regarded as the first step towards the redesign of propellers blade that of course will take advantage of the unloading provided by the higher thrust delivered by the newly designed duct.

3.1 Accelerating ducts

The outcomes of the design by optimization of the accelerating duct are summarized in the Pareto diagram of Figure 6. Among the design activities taken into consideration, the one involving the accelerating duct is the more complex. It considers, indeed, multiple functioning conditions. The nozzle is designed to provide the maximum thrust close to bollard pull condition that in numerical calculations was approximated with the functioning at a relatively...
low propeller advance coefficient ($C_{T \text{ prop.}} = 12$, $J_{\text{prop.}} = 0.3$) to avoid the complications related to zero inflow. At the same time minimization of the nozzle drag at higher advance coefficients ($C_{T \text{ prop.}} = 1.23$, $J_{\text{prop.}} = 0.8$) was required, together with maximization of inner static pressure (average pressure on a propeller disc in front of the actuator disc at $x/D = -0.15$ as in Figure 3) at both functioning. Results of the optimization process show certain margins of improvements, which can be verified by considering some relevant geometries among those belonging to the Pareto Frontier.

![Figure 6](image_url)

**Figure 6**: Pareto diagram of the accelerating duct design by optimization. Correlation between nozzle thrust and average inner pressure at the higher advance coefficient ($J = 0.80$, $C_{T \text{ prop.}} = 1.23$).

As outlined in the discussion, without redesigning the propeller, the recommended optimal nozzles are those operating at constant thrust coefficient in order not to change the functioning point of the propulsion system. Depending on the importance of bollard pull with respect to routinely operative conditions at higher advance coefficient, there are multiple possible choices. In present design, where the reference propeller/nozzle system was designed for take home purposes (i.e. provide a reasonably high thrust with a small propeller at a relatively high advance coefficient), it was considered mandatory, at first, to improve the performance at the higher advance coefficient. In order to limit cavitation phenomena, moreover, the increase of the inner nozzle pressure in this condition is another important objective of the design that turns into the second most important aspect for the selection of an optimal geometry. Among the Pareto geometries, ID 2172 is the optimal balance between the opposite design requirements. Providing a constant thrust at the higher advance coefficient ID 2172 increases, at that condition, the inner static pressure at its maximum without worsening the performance at bollard pull (constant thrust) in correspondence to which a certain improvement in terms inner pressure can be appreciated too. Other geometries presuppose trade-offs among objectives. At constant nozzle delivered thrust for $J = 0.80$, ID 1987 provides the maximum increase of static pressure (appreciable also at bollard pull) at the cost of a sensible reduction of bollard pull thrust. On the opposite, ID 1736 maximizes bollard pull performance at constant thrust for $J = 0.80$ with a detrimental influence on cavitation...
avoidance. By accepting the redesign of the propeller, ID 1835 exploits the maximum thrust from the propeller without any significant influence on pressure fields while ID 1056 is devoted only to the maximization of nozzle delivered thrust regardless the inner pressure field. Detailed descriptions of the geometrical modifications necessary for these achievements are summarized in Table 1 while in Figure 7 the computed pressure and velocity fields are shown for the selected geometries.

**Table 1:** Performance of the selected designs for the accelerating duct case.

<table>
<thead>
<tr>
<th>Nozzle</th>
<th>ID 19A</th>
<th>ID 2172</th>
<th>ID 1987</th>
<th>ID 1736</th>
<th>ID 1835</th>
<th>ID 1056</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_T$ Nozzle ($J = 0.3, C_T^\text{prop.} = 12$)</td>
<td>6.731</td>
<td>6.725</td>
<td>5.829</td>
<td>7.370</td>
<td>7.080</td>
<td>7.510</td>
</tr>
<tr>
<td>$C_T$ Nozzle ($J = 0.8, C_T^\text{prop.} = 1.23$)</td>
<td>0.238</td>
<td>0.236</td>
<td>0.237</td>
<td>0.237</td>
<td>0.267</td>
<td>0.282</td>
</tr>
<tr>
<td>$C_P$ ($J = 0.8, x/D = -0.15$)</td>
<td>-1.019</td>
<td>-0.861</td>
<td>-0.815</td>
<td>-1.096</td>
<td>-1.010</td>
<td>-1.278</td>
</tr>
</tbody>
</table>

**Figure 7:** Streamlines and inner pressure distribution for the selected optimal geometries. Accelerating nozzle at higher advance coefficient condition ($J = 0.8, C_T^\text{prop.}= 1.23$).

### 3.1 Decelerating ducts

The design of the decelerating nozzles has been carried out considering, in addition to the reference propeller load, two further conditions characterized by the same radial force distribution but total load respectively increased (+20%) and decreased (-20%). Aim of the analyses is to show the sensitivity of the nozzle design to the propeller functioning and consequently, the necessity of tailored designs. The results of the optimization activities are summarized in the Pareto diagram of Figure 8, where only the Pareto frontier for the three loading conditions are compared and the “optimal” designs for each case (maximized inner pressure at constant nozzle force and, vice versa, maximized nozzle force at constant inner pressure) are highlighted. Also the performance of the reference nozzle geometry operating in correspondence of the three loads are shown. Increasing the propeller load turns into a worsening of the inner pressure at almost constant nozzle drag. In the case of the reference geometry (similarly also for the optimized) the stagnation point moves to the outer surface of the nozzle (Figure 9) in the case of higher loads which imply a stronger contraction of the streamlines and a resulting “negative” angle of attack. Optimization allows certain margins for what regards both the maximization of the inner pressure and the reduction of the nozzle drag. In the worst case of the high propeller load (+20%), ID 2187 allows for positive values of inner pressure reducing the risk of cavitation with respect to the undisturbed flow. Similarly also a reduction of the nozzle drag is possible, regardless the propeller loading. In this case, however, the higher improvements are achieved with the high propeller load and ID 2182 ensures the higher relative improvement in terms of forces (Table 2) with respect to all
the others optimal configurations identified with different propeller loadings.

Figure 8: Comparison of the Pareto frontier for different propeller loading distributions.

Table 2: Performance of the selected designs for the decelerating duct case.

<table>
<thead>
<tr>
<th>Ref.</th>
<th>Ref. (+20%)</th>
<th>Ref. (-20%)</th>
<th>ID 2201</th>
<th>ID 2192</th>
<th>ID 2187</th>
<th>ID 2182</th>
<th>ID 2020</th>
<th>ID 1492</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_T$ / $C_T_{ref.}$</td>
<td>-0.018</td>
<td>-0.0187</td>
<td>-0.0173</td>
<td>-0.0182</td>
<td>-0.0149</td>
<td>-0.0149</td>
<td>-0.0173</td>
<td>-0.0146</td>
</tr>
<tr>
<td>$C_p$</td>
<td>0.035</td>
<td>-0.041</td>
<td>0.111</td>
<td>0.0861</td>
<td>0.0340</td>
<td>0.0239</td>
<td>-0.036</td>
<td>0.164</td>
</tr>
</tbody>
</table>

Figure 9: Streamlines and inner pressure distribution for the selected optimal geometries. Decelerating nozzle.
At constant inner pressure, the drag of the nozzle can be reduced up to 20%, which may represent a certain margin for the redesign of a propeller in a more favorable, for cavitation inception, unloaded functioning condition. Independently from the loadings, the optimal shapes are shorter and thinner [5,6], demonstrating the need of appropriate structural constraints for real and feasible applications. The suction peak on the outer duct surface is not sensibly affected by the geometrical modifications and the risk of sheet cavitation on the nozzle, even though not directly monitored throughout the optimization process, is not appreciably enlarged for the selected geometries.

4 CONCLUSIONS

In this paper, an optimization approach for the design of the nozzles for ducted propeller applications has been proposed. The design workflow, consisting in a parametric description of nozzle shape, in a fully scriptable mesh generation tool based on the OpenFOAM blockMesh library, in the efficient RANSE solver simpleFOAM and in an automatic post-processing of the data together combined through the ModeFrontier optimization environment, was applied for the design of both accelerating and decelerating ducts. The numerical model employed a simplified actuator disk with radially varying momentum sources in order to achieve the necessary computational efficiency required by a design through optimization. Common accelerating nozzles were designed for highly (tip) loaded propellers while for decelerating type shapes the availability of design guidelines is even more scarce, preventing their reliable application to different cases, such tip unloaded blades or lightly loaded conditions. The optimization environment meets these requirements: provide a reliable tool to customize the nozzle shape based on different objectives and constraints, allowing for the fully exploitation of the potentialities of ducted propellers. In the case of the accelerating duct it was possible to identify a set of trade-offs configurations for the multiple functioning conditions simultaneously addressed in the optimization. Improving the cavity inception speed was possible without any detrimental effects on thrust at both bollard pull and at the higher advance coefficient selected to resemble the routinely functioning of the propeller. A first step towards the combined design of the whole propulsive system was proposed in the case of the decelerating nozzle by analyzing designs for three different propeller loading. Also in this case, the outcomes of the design by optimization were encouraging. It was demonstrated that a dedicated nozzle design, as in the previous case, supports simultaneously the increase of the inner pressure and the reduction of drag, showing the benefits of a custom design based on actual requisites.

A redesign of the propeller geometry based on the newly devised nozzle shapes would give an insight into the possibility to improve the overall propulsive performance by using a propeller operating in a more favorable inflow, with a higher static pressure or in correspondence to a different functioning point thanks to the reduction of the nozzle drag. Definitely, a feedback mechanism would be necessary in order to account for the mutual influence between the propeller and the duct and carry out a combined design. The goal, finally, would be the simultaneous design of the nozzle and of the propeller by employing the well-established procedure for propeller optimization including the systematic variation of the duct driven by high-fidelity viscous calculations.
REFERENCES


WEC PARAMETERS OPTIMIZATION BY GENETIC ALGORITHM METHOD
MARINE 2017

A.JABRALI¹, R.KHATYR² AND J.KHALID NACIRI³
Laboratory of Mechanics, Faculty of Sciences Aïn Chock, B.P 5366, Maarif
Hassan II University, Casablanca, Morocco

Corresponding authors: E-Mail: (¹) jabraliahmed10@gmail.com
(²) khatyrrabha@gmail.com
(³) naciriuh2c@gmail.com

Key words: Genetic algorithm, parameters optimization, floating WEC, multi-body articulated system

Abstract. This study presents a method for parameters optimization for a floating wave energy converter (WEC) device. The considered floating WEC, a multi-body articulated system, consists of two cylinders connected with a flat plate. The connections between the parts of the WEC allow the rotational movements of cylinders and of the plate and the entire system perform translational movements. This study focuses on the case of two-dimensional movements of the WEC due to the action of waves which propagate perpendicular to the axis of the cylinders. The pressure and the viscous forces acting on the wetted surfaces of the cylinders are modeled by the Morison force equation [10], to which are added Archimedes and gravity forces.

The Newton laws written for the multi-body articulated system, whose movements have five degrees of freedom, result in a system of five nonlinear second-order differential equations which is solved numerically by a fourth order Runge-Kutta method [11]. The results show the effects of various parameters as the radius of the cylinders, the length of the relating plate, the coefficients of the power take off device, and the wave characteristics on the efficiency of the wave energy converter. To optimize these parameters values, we use a genetic algorithm method [7] for determination of optimal values. The first test of the method is an optimization of the power recovery coefficients for fixed values of geometric WEC parameters and of the wave characteristics. Thereafter, the genetic algorithm method is used to optimize various WEC parameters.
1 INTRODUCTION

Many wave energy recovery technologies have been developed in recent decades. A review of these technologies can be found in the refs. [1-3]. Among the various existing devices, the floating devices are of particular interest insofar as they do not require costly and complex fixing systems for their exploitation. High performance floating energy recovery devices such as PELAMIS [4], or SEAREV [5] systems are undergoing extensive work and are continually being improved by addressing key factors such as power recovery systems (PTO), shapes used, and other parameters.

The optimization of all the characteristic parameters of these systems is of great interest for their future development. Banos et al. [6] presents a review of numerical optimization methods used in the field of renewable energies. Genetic algorithms are tools which are promising for the optimization of floating energy converters insofar as they make it possible to envisage, at least at the theoretical level, an overall optimization of the device by acting on both the shapes and the characteristic parameters of the system in terms of magnitude and this in relation to the waves to which the converter is exposed.

This approach was adopted by Babarit et al. [7] who used the genetic algorithm to optimize the shape and mechanical parameters of the SEAREV device in order to maximize the annual production of energy at a given site. Similarly, McCabe et al. [8] studied the optimization of the shape of a wave energy collector to improve energy extraction using genetic algorithms. Recently, Zhang et al. [9] studied a multi-pendulum energy converter. The final structural parameters of the pendulum were obtained using a genetic algorithm based on the results of the numerical simulation of the pendulum structure.

This study presents a method for parameters optimization for a floating wave energy converter (WEC) device. The considered floating WEC, a multi-body articulated system, consists of two cylinders connected with a flat plate. The connections between the parts of the WEC allow the rotational movements of cylinders and of the plate and the entire system perform translational movements. This study focuses on the case of two-dimensional movements of the WEC due to the action of waves which propagate perpendicular to the axis of the cylinders.

The pressure and the viscous forces acting on the wetted surfaces of the cylinders are modelled by the Morison force equation [10], to which are added Archimedes and gravity forces.

The Newton laws written for the multi-body articulated system, whose movements have five degrees of freedom, result in a system of five nonlinear second-order differential equations governing the motion of the WEC which is solved numerically by a fourth order Runge-Kutta method [11]. The results show the effects of various parameters as the radius of the cylinders, the length of the relating plate, the coefficients of the power take off device, and the wave characteristics on the efficiency of the wave energy converter. To optimize these parameters values, we use a genetic algorithm method for determination of optimal values. The first test of the method is an optimization of the power recovery coefficients for fixed values of geometric WEC parameters and of the wave characteristics. Thereafter, the genetic algorithm method is used to optimize various WEC parameters.
2 MATHEMATICAL MODELING

In a Non-inertial reference frame $\mathcal{R}(O, \vec{x}, \vec{y}, \vec{z})$, where $O$ is an arbitrary point taken at the moving free surface of the fluid and $\vec{y}$ is the upward vertical, we consider the plane motion of an articulated multi-body system used as wave energy converter (WEC) and oscillating under the action of sea waves. The WEC consists of two cylinders, of centers $O_1$ and $O_2$ and radius $R_1$ and $R_2$ respectively, connected by a flat plate of center $G$ and length $L$. Taking into account the connections between parts of the system as shown in figure 1, and since only plane movements of the WEC are considered, we introduce five degrees of freedom for the mechanical system which are the heave ($y_1$), the surge ($x_1$) and the pitch ($\alpha_1$) for cylinder 1, the pitch ($\alpha_2$) for cylinder 2 and the angle ($\alpha$) for the plate. Here $x_1$, $y_1$ are the cartesian coordinate of $O_1$ in the frame $\mathcal{R}(O, \vec{x}, \vec{y}, \vec{z})$, $\alpha_1$ (resp. $\alpha_2$) is the angle between $\vec{x}$ and $\vec{x}_1$ (resp. $\vec{x}_2$) where $\vec{x}_i$ ($i=1,2$) is the axis of the relative frame of reference $\mathcal{R}_i(O_i, \vec{x}_i, \vec{y}_i, \vec{z})$ attached to cylinder $i$ and $\alpha$ is the angle between $\vec{x}$ and the plate $O_1B$.

\[
\begin{bmatrix}
D_i \\
\end{bmatrix} = \begin{bmatrix}
[r_{pl}] \\
[r_{Mt}] \\
[r_{Al}] \\
[r_{loit}] \\
[r_{vt}] \\
\end{bmatrix} - \begin{bmatrix}
[r_{el}] \\
\end{bmatrix}
\]

(1)

Where $[D_i]$ is the dynamic torsor, $[r_{pl}]$ represents the gravity force torsor, $[r_{Mt}]$ is the Morison force torsor representing the inertia forces and viscous forces exerted by the fluid on the system, $[r_{Al}]$ is the Archimedes thrust torsor, $[r_{loit}]$ represents the reactions torsor at connection between the cylinder $i$ and the plate, $[r_{vt}]$ is the forces torsor for the power take off system of the WEC and $[r_{el}]$ is the inertia force torsor related to the non-inertial character of
the considered reference frame. Since all the torsors are expressed at point \( O_i \), then the terms of equation (1) are given by:

\[
[D_i] = \begin{pmatrix}
    \frac{m_i\ddot{\bar{x}}_i}{2}
    \\
    \frac{m_i\ddot{\bar{y}}_i}{2}
    \\
    0
    \\
    0
    \\
    \frac{m_i\ddot{\bar{\alpha}}_i}{2}
\end{pmatrix},

[r_{pi}] = \begin{pmatrix}
    0
    \\
    0
    \\
    \tau_{Mi}
    \\
    0
    \\
    \tau_{Ai}
\end{pmatrix},

[r_{Loi}] = \begin{pmatrix}
    F_{ix}
    \\
    F_{iy}
    \\
    L_i
    \\
    M_i
    \\
    -\beta_i(\ddot{\bar{\alpha}}_i - \ddot{\bar{\alpha}})
\end{pmatrix}.

Were \( m_i \) is the mass of the cylinder \( i \), \( \ddot{\bar{x}}_i \) and \( \ddot{\bar{y}}_i \) are the two accelerations along \( \bar{O}_x \) and \( \bar{O}_y \) axis respectively, \( \ddot{\bar{\alpha}}_i \) is the angular acceleration of the cylinder \( i \), \( g \) represent acceleration of gravity, \( F_{mix} \) and \( F_{miy} \) are given by Morison equation [10], \( F_{arix} \) and \( F_{ary} \) are the two forces of Archimedes along the axis \( \bar{O}_x \) and \( \bar{O}_y \) respectively, defined by \( -\rho L_\nu \bar{\alpha} g \), where \( V_\nu \) is the immersed volume, \( F_{ix} \) and \( F_{iy} \) the reaction forces at the point \( O_i \) and \( B \) along the axis \( \bar{O}_x \) and \( \bar{O}_y \) respectively, \( L_i \) and \( M_i \) the reactions momentum at the point \( O_i \) and \( B \) along the axis \( \bar{O}_x \) and \( \bar{O}_y \) respectively, \( \beta_i \) is the damping coefficients related to the power take off devices, \( \ddot{\bar{\alpha}}_i \) is the angular velocity of the cylinder \( i \) and \( \ddot{\bar{\alpha}} \) is angular velocity of the plate, \( \eta \) represent the distance between the assumed flat bottom and the free surface.

For the plate the Newton's second law of motion is written as:

\[
[D_p] = [r_{pb}] - [r_{Lo1}] - [r_{Lo2}] - [r_{eb}]
\]

Where \( [D_i] = \begin{pmatrix}
    \frac{m_p\ddot{\bar{x}}_p}{2}
    \\
    \frac{m_p\ddot{\bar{y}}_p}{2}
    \\
    0
    \\
    0
    \\
    \frac{m_p\ddot{\bar{\alpha}}_p}{2}
\end{pmatrix},

[r_{pb}] = \begin{pmatrix}
    0
    \\
    0
    \\
    \tau_{L01}
    \\
    0
    \\
    \tau_{L02}
\end{pmatrix},

[r_{Lo1}] = \begin{pmatrix}
    0
    \\
    0
    \\
    \tau_{L02}
    \\
    0
    \\
    \tau_{L01}
\end{pmatrix},

[r_{Lo2}] = \begin{pmatrix}
    0
    \\
    0
    \\
    \tau_{Lo2}
    \\
    0
    \\
    \tau_{Lo1}
\end{pmatrix}.

with \( \eta \) represent the distance between the assumed flat bottom and the free surface, \( m_p \) is the mass of the plate, \( \ddot{\bar{x}}_p \) and \( \ddot{\bar{y}}_p \) are the two accelerations along \( \bar{O}_x \) and \( \bar{O}_y \) axis respectively, \( \ddot{\bar{\alpha}}_p \) is the angular acceleration of the plate, \( F_{1x} \) and \( F_{1y} \) the connection forces at the point \( G \) relative to \( O_1 \) along the axis \( \bar{O}_x \) and \( \bar{O}_y \) respectively, \( L_{G1} \) and \( M_{G1} \) the reaction moment at the point \( G \) relative to \( O_1 \) along the axis \( \bar{O}_x \) and \( \bar{O}_y \) respectively, \( F_{2x} \) and \( F_{2y} \) the reaction forces at the
point $G$ relative to $B$ along the axis $\overrightarrow{Ox}$ and $\overrightarrow{Oy}$ respectively, $L_{G2}$ and $M_{G2}$ the reaction moment at the point $G$ relative to $B$ along the axis $\overrightarrow{Ox}$ and $\overrightarrow{Oy}$ respectively.

By inserting the expressions of the torsors in equations (1) and (2), and after a rearrangement, we obtain the following system of five coupled differential equations for the five degrees of freedom $x_1$, $y_1$, $\alpha$, $\alpha_1$ and $\alpha_2$:

\begin{align}
M\ddot{x}_1 - m'L\sin \alpha \dot{a} - m'L \cos \alpha \ddot{a}^2 + m_2R_2 \cos \alpha_2 \ddot{a}_2 - m_2R_2 \sin \alpha_2 \dot{a}_2^2 + F_{m_2x} + F_{m_2x} &= 0 \\
-F_{ar_1x} - F_{ar_2x} &= 0 \\

M\ddot{y}_1 + m'L \cos \alpha \dot{a} - m'L \sin \alpha \ddot{a}^2 + m_2R_2 \sin \alpha_2 \ddot{a}_2 + m_2R_2 \cos \alpha_2 \dot{a}_2^2 + F_{m_2y} + F_{m_2y} = 0 \\
-F_{ar_1y} - F_{ar_2y} + Mg &= 0 \\

m'' \sin \alpha \dot{x}_1 - m'' \cos \alpha \dot{y}_1 + m''L \ddot{a} - m_2R_2 \sin(\alpha - \alpha_2) \ddot{a}_2 + m_2R_2 \cos(\alpha - \alpha_2) \dot{a}_2 \\
-(F_{ar_1x} - F_{ar_2x}) \sin \alpha + (F_{ar_1y} - F_{ar_2y}) \cos \alpha - m'' \cos \alpha g &= 0 \\

\dot{a}_1 + \frac{2}{m_1R_1^2} \beta_1(\dot{a}_1 - \dot{a}) = 0 \\

m_2R_2 \cos \alpha_2 \dot{x}_1 + m_2R_2 \sin \alpha_2 \dot{y}_1 - m_2R_2L \sin(\alpha - \alpha_2) \ddot{a} - m_2R_2L \cos(\alpha - \alpha_2) \dot{a}^2 \\
+ \frac{3}{2} m_2R_2 \ddot{a}_2 + \beta_2(\dot{a}_2 - \dot{a}) - R_2 \cos \alpha_2 F_{ar_2x} - R_2 \sin \alpha_2 F_{ar_2y} + m_2R_2 \sin \alpha_2 g &= 0
\end{align}

Where $M = m_1 + m_2 + m_b$ is the total mass of the WEC. Relations expressing Archimedes and Morison forces components are given in appendix 1. The numerical resolution of the coupled differential equations (3)-(7) is achieved by using 4th order Runge-Kutta method.

3 GENETIC ALGORITHM

In order to maximize the energy recovered by the device, the values of the characteristic parameters of the device such as the length of the plate and the damping coefficients of the power take off system are determined as functions of the characteristics of the waves. A genetic algorithm based code, presented in (Figure 2), is used to optimize the system. First test has been realized for the determination of the optimal values of the damping coefficients by leaving fixed the values of cylinder radius and wave characteristics and a comparison between the results obtained by genetic algorithm method and those of a direct calculation of
optimal values are achieved. After this validation tests, the damping coefficients and other parameters of the WEC geometry are determined in order to optimize the energy recovering.

**Figure 2**: Genetic algorithm for parametric optimization

### 4 RESULTS AND DISCUSSION

Figure 3 presents the direct calculation of the energy recovered for one period, depending on the coefficients $\beta_1$ and $\beta_2$ of the power take off devices of cylinder 1 and cylinder 2. It is noted that recovered energy increases up to optimum values of the two coefficients of PTO devices $\beta_1$ and $\beta_2$. 

```plaintext
4 RESULTS AND DISCUSSION

Figure 3 presents the direct calculation of the energy recovered for one period, depending on the coefficients $\beta_1$ and $\beta_2$ of the power take off devices of cylinder 1 and cylinder 2. It is noted that recovered energy increases up to optimum values of the two coefficients of PTO devices $\beta_1$ and $\beta_2$. 
```
Figure 3: The energy recovered depending to the PTO devices coefficients $\beta_1$ and $\beta_2$.

Figure 4 presents the convergence process for the genetic algorithm (G.A) method by plotting the best energy recovered at each generation versus the number of generations. In this case one presents the optimization of the parameters $\beta_1$ and $\beta_2$ of the PTO device where $R_1 = R_2 = 0.057m$ and $L = 0.15m$. It is noted that the optimal values of $\beta_1$ and $\beta_2$ obtained by G.A and by the direct calculation method are in close agreement (table 1). The advantage of using genetic algorithm method is to reduce the computer calculation time (see table 1).

Figure 4: The energy recovered depending to the generation.
Table 1: Time of calculation and recovered energy on one period.

<table>
<thead>
<tr>
<th></th>
<th>$\beta_1$ optimum</th>
<th>$\beta_2$ optimum</th>
<th>Energy (J)</th>
<th>Time of calculations (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>G.A</td>
<td>0.121</td>
<td>17.1</td>
<td>4.17</td>
<td>27000</td>
</tr>
<tr>
<td>Direct calculation</td>
<td>0.14</td>
<td>17</td>
<td>4.15</td>
<td>83056</td>
</tr>
</tbody>
</table>

In the second test, we optimized by GA method the parameters $\beta_1$, $\beta_2$ and $L$ in situation where $A_m = 0.02 \text{ m, } \omega = 0.2 \text{ rad/s and } R_1 = R_2 = 0.057 \text{ m}$, the results are satisfactory. It is noted that the new value of $L=0.16$ obtained by G.A increase the recovered energy to the value of 4.19J.

REFERENCES


ACKNOWLEDGEMENTS

The authors thank the reviewers for their valuable comments and suggestions that helped to improve the quality of the manuscript.

A. JABRALI, R. KHATYR AND J. KHALID NACIRI


APENDIX 1

ARCHIMEDES FORCE:

\[ F_{ar1x} = \rho_e g L_1 R_1 y_1 \left( \cos \left( \frac{\eta_1}{R_1} \right) - \cos \left( \frac{\eta_2}{R_1} \right) \right) - \frac{\rho_e g L_1 R_1^2}{4} \left( \cos \left( 2 \left( \frac{\theta_L + \frac{\eta_2}{R_1}}{2} \right) \right) + \cos \left( \frac{\theta_L + \frac{\eta_2}{R_1}}{2} \right) \right) \]  
\[ (A1-1) \]

\[ F_{ar1y} = \frac{\rho_e g L_1 R_1^2}{2} \left( \frac{\eta_2 + \eta_1}{R_1} + 2 \theta_L \right) + \rho_e g L_1 R_1 \left( \sin \left( \frac{\theta_L + \frac{\eta_2}{R_1}}{2} \right) + \sin \left( \frac{\theta_L + \frac{\eta_2}{R_1}}{2} \right) \right) \left( \frac{R_1}{4} - y_1 \right) \]  
\[ (A1-2) \]

\[ F_{ar2x} = \rho_e g L_2 R_2 \left( y_1 + L \sin(\alpha) - R_2 \cos(\alpha_2) \right) \left( \cos \left( \frac{\theta_{Lz} + \frac{\eta_3}{R_2}}{2} \right) - \cos \left( \frac{\theta_{Lz} + \frac{\eta_4}{R_2}}{2} \right) \right) \]  
\[ (A1-3) \]

\[ F_{ar2y} = \rho_e g L_2 R_2 \left( \sin \left( \frac{\theta_{Lz} + \frac{\eta_3}{R_2}}{2} \right) + \sin \left( \frac{\theta_{Lz} + \frac{\eta_4}{R_2}}{2} \right) \right) \left( \frac{R_2}{4} - y_1 - L \sin(\alpha) + R_2 \cos(\alpha_2) \right) \]  
\[ + \frac{\rho_e g L_2 R_2}{2} \left( \frac{\eta_2 + \eta_3}{R_2} + 2 \theta_{Lz} \right) \]  
\[ (A1-4) \]

With:

\[ \theta_L = \arccos \left( \frac{y_1}{R_1} \right), \quad \theta_{Lz} = \arccos \left( \frac{y_1 + L \sin(\alpha) - R_2 \cos(\alpha_2)}{R_2} \right), \]

\[ \eta_1 = A_m \sin(\omega t - k(x_1 - R_1 \sin(\theta_1))), \]

\[ \eta_2 = A_m \sin(\omega t - k(x_1 + R_1 \sin(\theta_1))), \]

\[ \eta_3 = A_m \sin(\omega t - k(x_2 - R_2 \sin(\theta_{Lz}))), \]

\[ \eta_4 = A_m \sin(\omega t - k(x_2 + R_2 \sin(\theta_{Lz}))), \]

\[ A_m \] is the amplitude of the wave of pulsation \( \omega \) and wave number \( k \).

MORISON FORCE:

\[ F_{mix} = \rho_e C_m V_i \ddot{x}_i + \frac{1}{2} \rho_e C_d S_i \dot{x}_i \left| \dot{x}_i \right| \]  
\[ (A1-5) \]

\[ F_{miy} = \rho_e C_m V_i \ddot{y}_i + \frac{1}{2} \rho_e C_d S_i \dot{y}_i \left| \dot{y}_i \right| \]  
\[ (A1-6) \]

Where \( \dot{x}_i \) and \( \ddot{x}_i \) are respectively the speed and the acceleration of the cylinder \( i \) along \( \vec{Ox} \), \( \dot{y}_i \) and \( \ddot{y}_i \) are respectively the speed and the acceleration of the cylinder \( i \) along \( \vec{Oy} \), \( \rho_e \) represent fluid density, \( C_m \) represent added mass coefficient, \( C_d \) is defined as drag
coefficient, \( S_i = R_i L_i \arccos \left( \frac{Y_i}{R_i} \right) \) is the wetted cross-section area of cylinder perpendicular to the direction of flow, \( L_i \) is the length of the cylinder \( i \), \( V_i \) is the volume of the body.
A NEW VOLUME-OF-FLUID METHOD IN OPENFOAM

Johan Roenby*, Bjarke Eltard Larsen*, Henrik Bredmose† AND Hrvoje Jasak‡

* DHI, Agern All 5, 2970 Hørsholm, Denmark, e-mail: jro@dhigroup.com
* DTU Mechanical Engineering, Richard Petersens Plads, 2800 Kgs. Lyngby, Denmark, e-mail: bjelt@mek.dtu.dk
† DTU Wind Energy, Technical University of Denmark, Nils Koppels Alle, 2800 Lyngby, Denmark
‡ Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb, Ivana Lucica 5, Zagreb, Croatia

Key words: CFD, Marine Engineering, Interfacial Flows, IsoAdvector, VOF Methods, Surface Gravity Waves

Abstract. To realise the full potential of Computational Fluid Dynamics (CFD) within marine science and engineering, there is a need for continuous maturing as well as verification and validation of the numerical methods used for free surface and interfacial flows. One of the distinguishing features here is the existence of a water surface undergoing large deformations and topological changes during transient simulations e.g. of a breaking wave hitting an offshore structure. To date, the most successful method for advecting the water surface in marine applications is the Volume-of-Fluid (VOF) method. While VOF methods have become quite advanced and accurate on structured meshes, there is still room for improvement when it comes to unstructured meshes of the type needed to simulate flows in and around complex geometric structures. We have recently developed a new geometric VOF algorithm called isoAdvector for general meshes and implemented it in the OpenFOAM interfacial flow solver called interFoam. We have previously shown the advantages of isoAdvector for simple pure advection test cases on various mesh types. Here we test the effect of replacing the existing interface advection method in interFoam, based on MULES limited interface compression, with the new isoAdvector method. Our test case is a steady 2D stream function wave propagating in a periodic domain. Based on a series of simulations with different numerical settings, we conclude that the introduction of isoAdvector has a significant effect on wave propagation with interFoam. There are several criteria of success: Preservation of water volume, of interface sharpness and shape, of crest kinematics and celerity, not to mention computational efficiency. We demonstrate how isoAdvector can improve on many of these parameters, but also that the success depends on the solver setup. Thus, we cautiously conclude that isoAdvector is a viable alternative to MULES when set up correctly, especially when interface sharpness, interface smoothness and calculation times are important. There is, however, still potential for improvement in the coupling of isoAdvector with interFoam’s PISO based pressure-velocity solution algorithm.
1 INTRODUCTION

Computational Fluid Dynamics (CFD) is quickly gaining popularity as a tool for testing and optimising marine structural designs and interaction with the surrounding water environment. A concrete example is the assessment of extreme wave loads on offshore wind turbine foundations of various types and shapes. From a numerical perspective, one of the great challenges within marine CFD is accurate description and advection of a complex free surface, or air-water interface. Various solution strategies have been developed to cope with this challenge[1]. The most widely used within practical interfacial CFD is the Volume-of-Fluid (VOF) method. In VOF, the interface is indirectly represented by a numerical field describing the volume fraction of water within each computational cell. The game of VOF is then about “guessing” how much water is floating across the faces between adjacent cells within a time step. VOF methods come in two variants: 1) geometric VOF methods, using geometric operations to reconstruct the fluid interface inside a cell and to approximate the water fluxes across faces, and 2) algebraic VOF methods, relying on the limiter concept to blend first and higher order schemes in order to retain sharpness and boundedness of the time advanced VOF field. Geometric VOF schemes are typically much more accurate, but also computationally more expensive, complex to implement, and restricted to certain types of computational meshes, such as hexahedral meshes. Algebraic VOF schemes, on the other hand, are less accurate, but often faster, easier to implement, and developed for general mesh types[2].

In marine applications, we often encounter complex geometries that are impossible, or at least very difficult, to represent properly with a structured mesh. Hence, most free surface CFD within marine engineering is based on algebraic VOF methods. Therefore, such simulations often require excessive mesh resolution and therefore long calculation times to obtain the desired solution quality.

To address the need for an improved computational interface advection method, we have developed a new VOF approach called IsoAdvector[3]. It is geometric of nature both in the interface reconstruction and advection step, but is applicable on general meshes consisting of arbitrary polyhedral cells. In the interface reconstruction, we take a novel approach using isosurface calculations to find the interface position and orientation in the intersected cells. In the advection step, we rely on calculation of the face-interface intersection line sweeping a mesh face during a time step. This avoids expensive calculations of intersections between cells and flux polyhedra[4]. For a thorough description of the isoAdvector concept the reader is referred to [3].

We have previously demonstrated using pure advection test cases that our new approach leads to accurate interface advection on both structured and unstructured meshes without compromising calculation times[3]. In OpenFOAM’s interfacial flow solver, interFoam, each time step starts by a MULES based update of the interface, followed by an update of the pressure and velocity, using a variant of the PISO algorithm[2]. In this segregated solution approach we can simply remove the MULES code snippet and replace it by a corresponding isoAdvector based snippet. To complete the replacement of MULES with isoAdvector, we must also calculate the mass flux across the faces – the quantity called rhoPhi in the interFoam code – based on isoAdvector, since this is needed in the convection term for the velocity field in construction and solution of the discretised momentum equations. In [5], we show how to derive a simple expres-
sion for rhoPhi from the mass fluxes provided by isoAdvector. The resulting solver is called interFlow and is provided as open source together with the isoAdvector library and various test cases at github.com/isoadvector.

In the following, we investigate the ability of interFlow and interFoam to propagate a stream function wave for 10 wave periods across a computational domain, which is exactly one wave length long and has periodic boundary conditions on the sides.

![Figure 1: The initial wave shape.](image)

Figure 1: The initial wave shape. The defining wave parameters are the water depth: $D = 20$ m, wave height: $H = 10$ m, wave period: $T = 14$ s and mass transport velocity: $\bar{u}_2 = 0$ m/s. Some derived quantities are the wave length: $L = 193.23$ m, steepness: $H/L = 0.052$, celerity: $c = 13.80$ m/s, crest height: $h_{crest} = 7.25$ m, crest particle speed: $u_{crest} = 5.95$ m/s, trough height: $h_{trough} = -2.75$ m, trough particle speed: $u_{trough} = -2.25$ m/s.

2 PHYSICAL SETUP

A stream function wave is a periodic steady wave calculated from potential flow theory using a truncated Fourier expansion of the surface and stream function describing the wave. The Fourier coefficients are calculated using a numerical root finding method in parameter space and by growing the wave height in steps so the solution procedure can be seeded with an Airy wave. For details on the solution procedure the reader is referred to [6]. Here we adopt the wave used in [7] and shown in Fig. 1, which also gives the wave parameters in the caption. The advantage of using stream function wave theory as opposed to Stokes N’th order theory for the input wave is that the former does not rely on the smallness of the wave amplitude, which is the Taylor expansion parameter of Stokes wave theory.

One thing to keep in mind, when attempting to reproduce potential theory waves in CFD is that these waves are calculated under the assumption of a free surface, i.e. zero pressure and no air phase on top of the water surface. In our simulation we have a second phase above the water and we are free to set the densities of the two phases. The water density will be set to 1000 kg/m$^3$. Ideally, we would like to set the air density to zero for our stream function test case, but numerical stability issues limit how low we can set the air density. We choose an air density of 1 kg/m$^3$, which is close to the real physical value. This is a good compromise, on one hand high enough to limit high density ratio related instabilities at the interface, and on the other hand low enough to make the air behaving like a “slave fluid” moving passively out of the way in response to motion of the much heavier water surface.

The viscosities in both phases is set to zero in accordance with potential flow theory and we have deactivated the turbulence model. This amounts to running the solver in “Euler equation
mode”, albeit the numerics will to some extend introduce an effective dissipation leading to a lack of strict energy conservation.

For waves on the space and time scales considered here surface tension is irrelevant and we set it to zero in our simulations.

3 NUMERICAL SETUP

The interFoam and interFlow solvers used in this study are based on the OpenFOAM-v1612+ version provided by ESI-OpenCFD. The details of the PISO algorithm implementation are described in [2] and can be studied by inspecting the OpenFOAM code library, which is freely available at openfoam.com.

For all simulations in the following the sides have periodic boundary conditions for both the VOF-field, \( \alpha \), the velocity field, \( U \), and the pressure, \( p \). On the top and bottom we have zero normal gradient for \( \alpha \) and \( p \), and a slip boundary condition for \( U \).

The mesh type with square cells and two refinement zones covering the interface region is show in Fig. 1. This is the finest mesh used in this study with 20 cells per wave height and 384 cells per wavelength in the interface region. Two coarser meshes with square cells were also used: One with the finest refinement removed, yielding 10 cells per wave height, and a very coarse mesh with no refinement at all and only 5 cells per wave height.

In all simulations we use adaptive time stepping based on a maximum allowed CFL number. We show results with CFL = 0.1, 0.2 and 0.4. It should be noted that in water wave simulations with interFoam the velocities in the air phase above the water surface are often higher than the maximum velocities in the water volume. The air behaviour depends a lot on the choices of numerical schemes and settings, but for our density ratio of 1:1000, it is not uncommon to see air velocities that are 2-3 times higher than the velocity of the water particles in the wave crests. Thus, in a simulation with a maximum allowed CLF number of 0.1 the actual maximum CFL number in the water volume may in fact stay below 0.05. It might be fruitful to introduce in the time stepping algorithm a separate CFL limit for each of the two phases.

Besides mesh and time resolution, the accuracy of wave propagation simulations depends on the choices of schemes for the different terms in the momentum equations. In particular the results are sensitive to the choice of time integration scheme. Therefore, in what follows, we show results for both pure Euler time integration and a 50% mixture of Euler and Crank-Nicolson. Another influential scheme is the momentum convection scheme. The convective term is linearized and treated implicitly, so we use the face mass fluxes from a previous time step or iteration to advect the updated velocity field through the face. For the cell-to-face interpolation involved in the discretisation of the convective term we use a TVD method specialised for vector fields, called limitedLinearV in OpenFOAM terminology. This scheme requires specification of a coefficient in the range \( \psi \in [0,1] \), where 0 gives best accuracy and 1 gives best convergence[8]. In the following we use \( \psi = 1 \).

All discretisation schemes and solver settings used in the presented simulations are listed in Appendix A and B to allow the reader to verify our results.
4 RESULTS

In the subsequent two sections we first vary the spatial resolution and then the CFL number to investigate its effect on the wave propagation properties of interFlow (isoAdvector) and interFoam (MULES).

![Figure 2: Surface elevation after 10 wave periods with CFL = 0.1. For convenience of plotting the horizontal axis has been compressed by a factor of 10. Black: Exact, Green: H/dx = 5, Blue: H/dx = 10, Red: H/dx = 20. Top panels: Euler time discretisation. Bottom panels: 50% blended Crank-Nicolson and Euler time integration. Left panels: interFlow/IsoAdvecto. Right panels: interFoam/MULES.](image)

4.1 Mesh refinement study

To investigate the effect of spatial resolution we simulate for L/dx = 5, 10 and 20 the propagation of the wave through the periodic domain for 10 wave periods (140 s) and plot the final surface curve compared to the exact theoretical solution. The results are shown in Fig. 2. We observe that:

- In terms of surface shape preservation the best performance is obtained with isoAdvector on the finest mesh where MULES gives a wiggly surface.
- In spite of the wiggly surface MULES is superior in terms celerity on the finest mesh with
almost no visible phase shift.

- Using isoAdvector on the coarsest mesh leads to excessive decay in wave height.
- On the intermediate mesh isoAdvector also has excessive wave height decay with Euler but not with Crank-Nicolson.
- MULES with Euler looks surprisingly good on the coarsest mesh. Inspection of the time series reveals that this is a “lucky” snapshot right after the wave has broken due to excessive steepening. In general it can not be recommended to use meshes with only 5 cells per wave height with the numerical setup used here.

In Table 1 we show the time it took for the simulation of the 10 periods to finish on a single core for the different combinations of schemes and resolutions. isoAdvector is significantly faster than MULES for all combinations except for the H/dx = 10 with Euler. For the best settings, H/dx=20 and Crank-Nicolson, isoAdvector is 32% faster than MULES and slightly faster than the MULES-Euler combination.

<table>
<thead>
<tr>
<th>H/dx</th>
<th>isoAdvector</th>
<th>MULES</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>314</td>
<td>335</td>
</tr>
<tr>
<td>10</td>
<td>1892</td>
<td>1228</td>
</tr>
<tr>
<td>20</td>
<td>4356</td>
<td>5741</td>
</tr>
</tbody>
</table>

(a) Euler

<table>
<thead>
<tr>
<th>H/dx</th>
<th>isoAdvector</th>
<th>MULES</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>304</td>
<td>435</td>
</tr>
<tr>
<td>10</td>
<td>918</td>
<td>1669</td>
</tr>
<tr>
<td>20</td>
<td>5624</td>
<td>8151</td>
</tr>
</tbody>
</table>

(b) Crank-Nicolson 0.5

Table 1: Simulation times in seconds on a single core for 10 periods.

### 4.2 Time refinement study

As shown in [3], isoAdvector is accurate in pure advection test cases for CFL number up to 0.5. It is our experience that isoAdvector works well for such cases even for CFL numbers closer to (albeit not exceeding) 1. In [3] we also demonstrate how MULES requires CFL < 0.1 to converge. We would therefore hope that replacing MULES with isoAdvector in interFoam could allow more accurate solutions with larger time steps. In Fig. 3 we show the results of an exercise where we keep the mesh resolution fixed at H/dx = 20 and vary the CFL time step limit from 0.1 to 0.2 and on to 0.4. We observe that:

- isoAdvector with Euler gives excessive wave damping for CFL = 0.2 and 0.4.
- isoAdvector with Crank-Nicolson 0.5 gives slightly worse but acceptable results with CFL = 0.2 with an increase in phase error and overprediction of wave height.
- isoAdevctor with Crank-Nicolson and CFL = 0.4 causes severe wave breaking.
- MULES with Euler and CFL = 0.4 crashes before the simulation has finished.
- In spite of its wiggly surface MULES with CFL = 0.2 is very close to the CFL = 0.1 result only differing by a small phase error.
Figure 3: Surface elevation after 10 wave periods. For convenience of plotting the horizontal axis has been compressed by a factor of 10. Black: Exact, Green: CFL = 0.4, Blue: CFL = 0.2, Red: CFL = 0.1. Top panels: Euler time discretisation. Bottom panels: 50% blended Crank-Nicolson and Euler time integration. Left panels: interFlow/IsoAdvector. Right panels: interFoam/MULES.

This is somewhat disappointing for our hopes that isoAdvector would allow accurate simulations with large time steps. It should be noted, that the current coupling of isoAdvector with the pressure-velocity coupling is the simplest possible. Probably one should look for an improvement in this coupling rather than for an improvement in the inner workings of the isoAdvector method itself.

4.3 Crest velocity profiles

An important feature to be able to capture accurately in wave propagation simulations is the particle kinematics in the wave crest. As for instance shown in [9], many solvers have issues with overshooting in the particle velocities in the top of the crest. To investigate this, we show in Fig. 4 the variation in the x-component of the velocity along a line of cells going up through the wave crest. The results are shown for the simulations with H/dx = 20 and CFL = 0.1 at time t = 70 s, i.e. after 5 wave periods. It is evident from this figure that with the current implementation of isoAdvector into interFoam we get higher overshoots in the crest velocities than the original interFoam solver with MULES which does a remarkably good job with the
Figure 4: Horizontal velocity in cell centres at wave crest after 5 wave periods of the simulation with $H/dx = 20$ and CFL = 0.1. Red: Exact, Green: Simulation result. The $\alpha$-field is shown in a black-white colour map and the $\alpha = 0.001$, 0.5 and 0.999 contours are plotted in blue. Top panels: Euler time discretisation. Bottom panels: 50% blended Crank-Nicolson and Euler time integration. Left panels: interFlow/IsoAdvector. Right panels: interFoam/MULES.

Crank-Nicolson 0.5 time integration. Since the surface is advected passively in the velocity field, one should think that there was a strong correlation between a solver’s ability to represent these velocities accurately near the surface and its ability to accurately propagate the surface and preserve its shape. This does not seem to be the case here where isoAdvector, in spite of its errors in crest kinematics, produces a better surface, and MULES, in spite of its accurate crest kinematics, produces a wrinkled surface.

In Fig. 4, we show the interface width by plotting the $\alpha = 0.001$, 0.5 and 0.999 contours in blue colour. Careful inspection reveals that the distance between the 0.001 and 0.999 contours with isoAdvector is 3 which is essentially the theoretical minimal interface width for a VOF method. The corresponding distance with MULES is approximately twice as large, i.e. approximately 6 cells. This moves the stagnation point, where the air velocity above the crest changes direction,
one cell closer to the surface. In a true two-phase potential flow solution the tangential jump in velocity should be right on the interface. In this sense the isoAdvect solution is closer to the theoretical one.

4.4 Cell aspect ratio

It has previously been shown that the cell aspect ratio can have a significant effect on the propagation of waves in OpenFOAM and on the breaking point of shoaling waves[10]. Clearly, independence of simulation results on cell aspect ratios and cell shapes in general are highly desirable features. To investigate how isoAdvect behaves with different cell aspect ratios we have repeated the simulations on a mesh with flat cells \( (H/dx = 10 \text{ and } H/dy = 20) \) and on a mesh with tall cells \( (H/dx = 20 \text{ and } H/dy = 10) \) in the interface region. The results are shown in Fig. 5 where they are compared to the finest resolution results shown previously. We see that the halving of the cell count in the interface region has only a small effect on the isoAdvect simulation results. For MULES the surface wrinkles are exacerbated when using tall cells. For flat cells the wrinkles completely disappear and a slight phase error is introduced.

![Figure 5: Surface elevation after 10 wave periods. For convenience of plotting the horizontal axis has been compressed by a factor of 10. Black: Exact. Red: square cells, \( H/dx = H/dy = 20 \). Blue: flat cells, \( H/dx = 10, H/dy = 20 \). Green: tall cells, \( H/dx = 20, H/dy = 10 \). Top panels: Euler time discretisation. Bottom panels: 50% blended Crank-Nicolson and Euler time integration. Left panels: interFlow/IsoAdvect. Right panels: interFoam/MULES.](image)

5 CONCLUSION

We have demonstrated the feasibility of using the new geometric VOF algorithm, isoAdvect, in the OpenFOAM interfacial flow solver, interFoam, to propagate a steady stream function wave through a periodic domain. The benefits of using interFlow (interFoam with isoAdvect) as opposed to MULES is a sharper and more smooth surface, shorter calculation times and less sensitivity to cell aspect ratio. It is not recommended to use the solver with Euler integration and fewer than 10 cells per wave height. Satisfactory results are obtained with a 50:50 blend of Euler and Crank-Nicolson and 20 cells per wave height.
In spite of problems with a wrinkly surface the original interFoam solver with MULES performs better than interFlow when it comes to phase error (celerity) on the finest mesh and reproduction of the theoretical crest kinematics profile. Also, at this stage interFlow does not produce satisfactory results when running with CFL number > 0.2 as one might otherwise hope due to its ability to advect interfaces accurately at CFL numbers close to 1. We expect that higher accuracy at larger CFL numbers can be obtained by improving the way isoAdvector is coupled with the PISO loop in the interFoam solver.

Finally a word of caution regarding this kind of numerical comparisons. Choosing schemes and solver settings is a delicate procedure which requires some degree of informed guessing. It may well be that one combination of schemes produces accurate results for a particular test case because the energy that, say, the chosen time integration scheme erroneously injects into the system is by pure luck equal to the energy erroneously taken out of the system due to the coarseness of the mesh. Results may then look reasonable even though the numerical calculation does not in reality represent the simulated physics properly. A professional CFD engineer should always stress test her setup with an attitude of trying to prove it wrong, rather than trying to prove it right.

Acknowledgements

This work was funded by JR’s Sapere Aude: DFF-Research Talent grant from The Danish Council for Independent Research | Technology and Production Sciences (Grant DFF-1337-00118) and by DHI’s GTS grant from the Danish Agency for Science, Technology and Innovation.

A Solver settings

PIMPLE
{
  momentumPredictor yes;
  nCorrectors 3;
  nOuterCorrectors 1;
  nNonOrthogonalCorrectors 0;
  nAlphaCorr 1;
  nAlphaSubCycles 1;
  cAlpha 1;
  pRefPoint (1 0 16);
  pRefValue 0;
}

"alpha.water.*"
{
  nAlphaCorr 2;
  nAlphaSubCycles 1;
  cAlpha 1;
}

isoAdvvector
{
  interfaceMethod isoAdvvector;
  isoFaceTol 1e-8;
  surfCellTol 1e-8;
  snapAlpha 1e-12;
  nAlphaBounds 3;
  clip true;
}

p_rgh
{
  solver GAMG;
  tolerance 1e-8;
  relTol 0.01;
  smoother DIC;
### Discretisation schemes

ddtSchemes{default CrankNicolson 0.5;} // Euler
gradSchemes{default Gauss linear;}
divSchemes
{
  div(rhoPhi,U) Gauss limitedLinearV 1;
  div(phi,alpha) Gauss vanLeer;
  div(phirb,alpha) Gauss interfaceCompression;
  div(((rho*nuEff)*dev2(T(grad(U))))) Gauss linear;
}
laplacianSchemes{default Gauss linear corrected;}
interpolationSchemes{default linear;}
snGradSchemes{default corrected;}

REFERENCES


THE DVH MODEL: SIMULATING 2D VISCOUS FLOWS AROUND FIVE DIFFERENT BODIES AT RE=10,000

Emanuele ROSSI∗, Andrea COLAGROSSI†, David LE TOUZÉ∗

∗ École Centrale Nantes, LHEEA research department (ECN/CNRS), Nantes, France
† CNR - INSEAN, Via di Vallerano, Rome, Italy

Key words: Viscous flow around blunt bodies, wing profile in stall condition, Particle Vortex Method, Diffused Vortex Hydrodynamic, vortical wakes.

Abstract. The Diffused Vortex Hydrodynamic (DVH) is a meshless 2D Lagrangian incompressible Particle Vortex Method. In this study the DVH will be applied to simulate viscous flows around five different bodies at Re=10,000. In order to show how the DVH method works, simulations about five different geometries will be discussed: (i) a cylinder with circular section, (ii) a cylinder with squared section, (iii) a narrow elliptical profile, (iv) a thick elliptical profile and (v) a NACA0010 profile. All the five considered geometries develop a complex vortical wake and, moreover, the last three profiles, having a 30 degrees angle of attack, give rise to flow separation on the suction side.

1 Introduction

The study describes an improved meshless 2D Lagrangian incompressible Particle Vortex Method called Diffused Vortex Hydrodynamic (DVH). Deeply described and validated ([1, 2, 3, 4]), the method will be briefly presented in the following section. The DVH method brings the following advantages:

(I) the continuity equation is automatically satisfied;

(II) the pressure field is no longer a direct unknown of the problem, thanks to the vorticity formulation of the Navier-Stokes equation;

(III) the vorticity formulation allows to discretize only the rotational region of the flow (self-adaptivity);

(IV) high accuracy on the velocity field evaluation (obtained through a spatial integration);

(V) boundary conditions at infinity are automatically satisfied;

(VI) the Lagrangian nature of the method, compared to mesh-based approaches, reduces the numerical dissipation due to Navier-Stokes equations non-linear term.
However, the Lagrangian nature of this method needs the introduction of redistribution techniques to preserve accuracy (see e.g. [5]).

The DVH method is coupled with a low CPU costs packing algorithm [6] which generate a “Regular Point Distribution” (RPD) close to a solid surface. DVH is a body-fitted method and thanks to the RDPs the velocity boundary condition are enforced exactly on the body surface, therefore.

This model is particularly suited to get a correct description of the vortical fluid motion around a body of complex geometry.

To show the DVH method ability in solving complex flows, five different body geometries at Re=10,000 will be discussed.

2 The DVH method.

The DVH is governed by the Navier-Stokes equation written in vorticity formalism:

$$\frac{D}{Dt} \omega = \nu \Delta \omega \quad \forall \mathbf{r} \in \Omega,$$

where $D/Dt$ is the time material derivative, $\omega$ is the vorticity, $\nu$ is the kinematic viscosity and $\Omega$ is the domain of the 2D viscous incompressible flow.

In order to solve eq. 1 at each time step the operator splitting scheme [7, 8] will be used. To obtain the numerical solution is necessary to solve an inviscid advective step governed by the Euler equation:

$$\frac{D}{Dt} \omega(\mathbf{r}, t) = 0$$

(2)

together with a purely diffusive step governed by the heat equation:

$$\frac{\partial}{\partial t} \omega(\mathbf{r}, t) = \nu \Delta \omega(\mathbf{r}, t).$$

(3)

The presence of a body within the flow field requires to add a specific process, between the advective and the diffusive steps, generating vorticity on the body contour in order to enforce the no slip boundary condition.

The vorticity field is discretised as a sum of $N$ vortices, each of them represented by a positive smooth approximation of the Dirac $\delta$ distribution, $\delta_{\epsilon}$, and a circulation $\Gamma_i$:

$$\omega(\mathbf{r}, t) = \sum_{i=1}^{N} \Gamma_i(t) \delta_{\epsilon}(\mathbf{r} - \mathbf{r}_i).$$

(4)

Before starting the simulation the Packing algorithm described in [6] is used to generate a set of points regularly distributed (RPD) around the solid body. This procedure allows to arrange points around complex contours, while preserving the volume around each point.

During the advective step the velocity field is decomposed as follows:

$$\mathbf{u} = \mathbf{u}_{\infty} + \mathbf{u}_{\omega} + \mathbf{u}'$$

(5)
where \( \mathbf{u}_\infty \) is the free stream velocity, \( \mathbf{u}_\omega \) is the velocity induced by the vortex particles, while \( \mathbf{u}' \) is the contribution due to the presence of the body within the flow. The velocity \( \mathbf{u}_\omega \) can be evaluated using the Biot-Savart law, written as:

\[
\mathbf{u}_\omega(r, t) = \sum_{i=1}^{N} \Gamma_i(t) \mathbf{K}_\epsilon(r, r_i)
\]  

(6)

where \( \mathbf{K}_\epsilon \) is the mollified Biot-Savart kernel.

To evaluate the term \( \mathbf{u}' \) the body is discretised using a set of sources and point vortices. From \( \mathbf{u}' \) it is also possible to generate a new set of vortices on the body surface, enforcing the no-slip boundary condition exactly on its contour.

A Fast Multiple Method (FMM) is used to speed up the evaluation of the velocity \( \mathbf{u} \).

The vortices are displaced using a fourth order Runge-Kutta algorithm.

The diffusive step is performed following the deterministic algorithm described in [9]: each vortex particle gives a vorticity contribution on the RPDs by a superposition of elementary solutions of the heat equation (see [1, 3]). The set of points created during the diffusion process become the new set of vortex particles overwriting the previous one. The use of RPDs during diffusion impedes the excessive clustering or rarefaction of the vortex particles and avoids the use of remeshing procedures.

In order to solve the diffusion near a smooth solid boundary, an homogeneous Neumann condition for the vorticity field, together with a flat plate approximation of the solid contour, will be used. This approximation is no longer valid if the body has geometrical singularities. In this case is necessary to introduce a suitable visibility mask as explained in [3].

Another characteristic of the DVH method is the multi-resolution approach. That implies using several overlapping RPDs, which spatial resolution decreases as the distance from the body increase (see [2]).

3 Flow around a circular cylinder at \( \text{Re}=10,000 \).

In the present section the DVH simulation of the flow past a circular cylinder at \( \text{Re}=10,000 \) will be discussed. The considered cylinder has a diameter equal to \( c \) and the Reynolds number is defined as \( \text{Re} = Uc/\nu \), where \( U \) is the free stream velocity.

As discussed in [10], in a two dimensional framework the flow for a cylinder at \( \text{Re}=10,000 \) remain at a lower sub-critical regime: the shear layers start to fluctuate and only few eddies are formed downstream. As visible in the left plot of figure 1, the first seconds of the simulation are characterized by the formation of two vortical structures in the recirculation areas behind the cylinder. It is also possible to notice the formation of small eddies in the near wake due to the shear layer dynamic. The two recirculation bubbles remain stable until \( tU/c = 10 \). After this time the wake becomes unstable and the shedding mechanism begins, in which shed eddies form dipole structures downstream in the wake. Moreover, center and right plots of Figure 1 depict the vorticity field in the near wake region at the minimum and maximum of the lift.

The shedding mechanism induces oscillations on the forces, more visible on the lift than on the drag. The time evolution of lift and drag coefficients are reported in figure 2. The latter are defined as:

\[
C_d = \frac{F_d}{(1/2 \rho U^2 c)} \quad C_l = \frac{F_l}{(1/2 \rho U^2 c)}
\]  

(7)
Figure 1: Flow past a cylinder at $Re=10,000$. Left: recirculation of the shear layer at the beginning of the simulation. Center: vorticity field near the body at minimum lift. Right: vorticity field near the body at maximum lift. The dimensionless vorticity $\omega_c/U$ scales from -40 to 40.

Figure 2: Flow past a cylinder at $Re=10,000$: lift and drag coefficients time histories.

Figure 3: Wake at the end of the simulation for the cylinder at $Re=10,000$. The dimensionless vorticity $\omega_c/U$ scales from -40 to 40
where $\rho$ is the fluid density.

At this Reynolds number the forces time behaviour is aperiodic. Because of the aperiodic behaviour of the shedding mechanism each eddies shed have a different intensity and the wake is irregular as shown in Figure 3.

### 4 Flow around a square cylinder at $Re=10,000$.

This section study the flow past a square cylinder at $Re=10,000$. The length of the side is indicated with $c$. The DVH method can solve highly complicated flows, as visible from the beginning of the simulation.

Starting the simulation, the flow separates at each of the four vertices, and that generates four thin shear layers, as shown in left plot of Figure 4.

The shear layers detaching from the front part of the body directly induce a backward flow on the two side of the square cylinder parallel to the free stream direction, causing a local change of sign in the vorticity field. As a consequence of the interaction of positive and negative vorticity on the horizontal faces of the cylinder, small eddies are generated and convected downstream. Moreover, the detached shear layers roll up at the back of the square, forming two recirculation bubbles. Once the front detached eddies cover the whole length of the square cylinder, no more shear layer will be detached from the back of the body, interrupting the growth of the recirculation bubbles, as in the center plot of Figure 4. Continuing the simulation, just small eddies are detached from the body, not only from the horizontal faces but also from the back one, as visible in right plot of Figure 4.

Because of the detaching of small eddies, no principal shedding mechanism can be observed as well as in the lift and drag coefficients time histories (Figure 5).

The absence of regular shedding of large eddies is visible also on the wake (Figure 6). The wake is mainly formed by small eddies, and, however, while these eddies move inside the wake they interact with each other merging and forming dipole structures. For this geometry the wake appears more compact in space than the wake shed of the circular cylinder (see Figure 3).

![Figure 4](image_url)

**Figure 4**: Simulation initial stages of the flow past a square cylinder at $Re=10,000$. The dimensionless vorticity $\omega_c/U$ scales from -40 to 40.
5 Flow around three different airfoils at Re=10,000 in stall condition.

In this section three different airfoils at Re=10,000 and angle of attack $\alpha = 30^\circ$ are considered:

(i) a thick elliptical profile with aspect ratio between minor axis, $b$, and the major axis, $c$, equal to $b/c = 0.25$;

(ii) a narrow elliptical profile with aspect ratio $b/c = 0.1$;

(iii) a NACA0010 profile with chord $c$.

The viscous flows around these three geometries at Re=10,000 are characterized by the formation of shear layers detaching from the leading and the trailing edge. At the considered viscosity level, the shear layers roll up mechanism takes place.

The shear layer detaching from the trailing edge could also become unstable during the roll-up, generating a series of small eddies, usually called secondary vortices, as shown in left plot of Figure 7.
Figure 7: From top to bottom: detail of the vorticity field near the airfoils and of the loads. Left: vorticity field near the airfoil with the formation of secondary vortices. Center: detail of the vorticity field with the vorticity structure responsible for the shedding of the secondary vortices. Right: detail of the loads during the shedding of the secondary vortices. The dimensionless vorticity $\omega_c/U$ in the vorticity fields plots scales from -40 to 40.
The mechanism of formation of these secondary vortices at Re=10,000 has been described in [11], [12] and [13]. The trailing edge anti-clockwise vortex, in some cases, could interact with the boundary layer on the suction side of the body. If this happened, a local acceleration of the flow, generating a small clockwise vorticity structure on the trailing edge, can be observed. The clockwise vorticity structure is unstable and with a periodical oscillation, leading to the fragmentation of the counter-clockwise shear layer.

The above mentioned mechanism is shown in Figure 7: in the left plots the fragmentation of the counter-clockwise shear layer is visible in all the considered geometries; in the central plots the specific behaviour on the trailing edge is highlighted.

Secondary vortices mechanism influences also lift and drag forces, introducing components with high frequency and low intensity, as reported in the right plots of Figure 7.

Moreover, time histories of the drag and lift coefficients are presented in left plots of Figure 8. The force time signals have a complex evolution with different frequency components and absence of periodicity.

As a results of the above mechanism the wake are more complex, that means that during the wake evolution dipole structures, generated by the leading and trailing edge shear layers, interact with secondary vortices.

The ability of the DVH to simulate long wakes with complicated vorticity field is well represented in right plot of Figure 9.

The wake is formed by a series of dipole structures travelling upwards together with secondary structures travelling in the free stream direction.

As reported in Figure 9, the thickness of the considered bodies can influence the vorticity field, for example, the thick ellipse wake is characterized by larger wave numbers (i.e. larger vortex scales), compared to the other two profiles.

6 Conclusions

The DVH method, as in [1, 2, 3, 10] as well as in this article, is particularly suited in dealing with complex and/or narrow body geometries with vertices, especially because of its ability to solve and get a correct description of the 2D vortical fluid motion around body of complex geometry.

The study focused on DVH simulations at Re=10,000. At this viscosity level the vortex dynamic in the wake region has a complex nature while on the other hand the Reynolds number is too low to allow the transition to turbulence regimes. In such conditions numerical simulations can be quite challenging because boundary layers as well as near wake regions need to be properly resolved. Thanks to its characteristics, the DVH can be applied to solve these kind of challenging Reynolds numbers flows.

Moreover, DVH can be a useful tool in analysing viscous regime, as previously discussed, and be of interest in the context of maritime engineering when considering, for example, control surfaces of unmanned underwater vehicles (UUV) (see e.g. [14, 15, 16]) or problem related to Vortex Induced Vibrations of Marine Risers (see e.g. [17, 18]).
Figure 8: From top to bottom: Lift and Drag coefficient time histories for the ellipse with aspect ratio 0.25, the ellipse with aspect ratio 0.1 and the NACA0010 respectively.
Figure 9: From top to bottom: vorticity field at the end of the simulation for the ellipse with aspect ratio 0.25, the ellipse with aspect ratio 0.1 and the NACA0010 respectively. The dimensionless vorticity $\omega c/U$ scales from -2 to 2.
Acknowledgements

This work has been partially supported by the Flagship Project RITMARE - Italian Research for the Sea - coordinated by Italian National Research Council and funded by Italian Ministry of Education, University and Research within Nat. Res. Program 2015-2016, and has received a post-doctoral grant from Ecole Central Nantes.

REFERENCES


CENTRIFUGE ROLLING TEST FOR ORE LIQUEFACTION ANALYSIS

L. THOREL, PH. AUDRAIN, A. NEEL, A. BRETSCHNEIDER, M. BLANC

French Institute of Science and Technology for Transport, Development and Networks (IFSTTAR), GERS Dept., Geomaterials & Modeling in Geotechnics Lab., F-44340 Bouguenais, France

Luc.Thorel@ifsttar.fr, ORCID 0000-0002-0218-4144

Key words: Physical modelling, Centrifuge, Ore Cargo, Liquefaction

Abstract. To study the development of liquefaction in ore cargo, a new Rolling Test has been designed to support similar stresses than those observed in a vessel. It can be used in an 80×g macrogravity field in the 5.5m radius Ifsttar geo-centrifuge. Its main characteristics are presented.

1 INTRODUCTION

Combination of cyclic loading, presence of fine particles and variable moisture content within a bulk carrier’s ore cargo can result in liquefaction causing the vessel to list or capsize and possibly loss of human life. Three elements may generate such a catastrophic event: the cargo properties, the ship design and the sea conditions. In order to investigate the origin of cargo liquefaction during transportation, a new device has been developed at Ifsttar in the framework of the Franco-German European LiquefAction project.

From a geotechnical point of view liquefaction is a hazardous phenomenon; it consists in a change of the soil behavior from “solid” to “liquid” (the meaning of the word “liquefaction” is different than in physics, where a change from gas phase to liquid phase is observed). This phenomenon is related to the presence of interstitial water which, under specific loading conditions, can generate overpressures on the soil grains, up to a level sufficient to eliminate the contacts between the grains. The liquefaction phenomenon occurs “rapidly”, it means that the overpressures generated (e.g. by compaction) cannot be dissipated due to a very rapid solicitation or to medium-low soil permeability. Liquefaction accidents are often observed during earthquakes and can for instance involve building foundations.

Ore cargoes liquefaction is a complex and not fully understood phenomenon, it is not necessarily similar to seismic liquefaction even if similar effects can be observed. It could be related to different hypotheses basically ascribable to cyclic loading, fluid migration, and soil compaction. Of course the type of material, loading conditions and water content are the main parameters that influence the triggering of this phenomenon. To identify the risk of cargo liquefaction, several tests can be found in the literature ([1], Erreur ! Source du renvoi introuvable.): flow table test (derived from [3]), penetration test, weight penetration test and rolling test. The latter consists in a 0.3×0.3×0.3m Perspex cubic box, which is rotated around
a horizontal axis located in the middle of the box base. The Rolling period simulated is 10s, the maximum rotation angle of rolling is ±25°, and the test duration is limited to 5 min. All those tests are focused on the identification of the Transportable Moisture Limit (TML) of the ore cargo. The TML is the maximum Moisture Content (MC) of the ore cargo for which there is no risk of “flow”. MC is linked to the water content w by: MC = w/(1+w).

Liquefaction in ore cargoes is still an open problem; the phenomenon is not yet totally understood, no observations being possible during the shipment. In this sense physical modelling is a useful tool to observe the evolution of the material during shipment by artificially reproduce conditions similar to those occurring during sailing.

2 EXPERIMENTAL SET UP

The Rolling test presented here has been designed to be used in a geotechnical centrifuge, which allows the reproduction of similar stresses and pressures on the sample than inside the vessel.

2.1 Geotechnical centrifuge

Centrifuge modelling of geotechnical structures (foundations, tunnels, etc.) using physical models requires respecting the scaling laws (e.g. [2],[5]) that correlate the (reduced scale) model, placed in a macrogravity field, with the (full-scale) prototype targeted by the simulation exercise. This method has become quite wide-spread [6] and enables conducting parametric studies, ultimately taking structures to their failure, observing and understanding phenomena and obtaining data that can be applied either for drawing comparisons with actual structures or for calibrating numerical models. One of the strengths of this approach lies in its compatibility with the stress and strain states between two similar (homologous) points on both the reduced-scale model and prototype structure. This condition must in fact be satisfied since the soil behaviour and, more generally, the behaviour of all granular materials depends to a great extent on the applied stress level.

The primary scaling laws have been listed on Figure 1, with N representing the macrogravity intensity (or g-level). As an example, an experiment conducted at N=80 corresponds to a macrogravity field of 80×g and to a reduced model scale of 1/80.

The centrifuge acceleration (ω² ∙R) depends on both the angular rotation speed ω and the radius R at which the model is positioned.

<table>
<thead>
<tr>
<th>Scaling laws</th>
<th>L*=1/N</th>
<th>g*=N</th>
<th>σ*=1</th>
<th>ε*=1</th>
<th>F*=1/N²</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length, displacement</td>
<td>L*</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Centrifugal, seismic</td>
<td>g*</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>acceleration</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 1 : 200×g-tons & 5.5m radius Ifsttar’s geo-centrifuge and main scaling laws
2.2 Rolling test device

The device has been developed in order to simulate a rolling movement of a ship that transports possible iron ore cargo. In this case, the grain density reaches up to 4.8, when the dry density is about 3.

Rolling box

The box itself (Figure 2) is designed similarly to the Rolling Test Equipment suggested by ClassNK (2012), but it has been reinforced in order to support the stresses induced by the macrogravity field. A transparent face allows the observation of phenomena occurring inside the box during the movement. Inside the box, the two other lateral walls (port and starboard sides) have been designed with special features that allow studying different boundary conditions (rough, smooth, drained…).

The box is fixed in a rotating cradle (radius = 400 mm), placed on an assembly including rolls and hydraulic rotative actuator. A cog-wheel, linked to the actuator, moves a rack and pinion fixed on the cradle. The axis of rotation of the box is perpendicular to the centrifuge rotation axis.

The elevation of the box (Figure 3) may be adjusted in order to simulate the rolling movement in different cases: when the centre of gravity is higher than the axis of rotation (light ship) or when the centre of gravity is lower than the axis of rotation (heavy cargo). The elevation is fixed before the test by adjusting wedges of different thicknesses.

The stresses induced in the cargo are similar to the ones existing in a vessel, as the maximum height of cargo is 0.3 m in the box, which corresponds to 24 m at a centrifuge acceleration of 80×g.
Hydraulic rotative actuator

The rotative actuator has been selected in order to apply the required torque in the worst condition, when the centre of gravity is less elevated than the centre of rotation, and also taking into account the required frequency.

The model selected is a Parker HTR45 hydraulic rotative actuator, which allows a torque of 2000 N·m and a maximum service pressure of 80 bar. It contains an oil volume 1.8 dm³, which required an oil debit of 21 dm³/min at 0.1 Hz, or 105 dm³/min at 0.5 Hz. Those performances require of course an adequate hydraulic power supplies by a high pressure hydraulic pump with a flow of more than 100 dm³/min.

Control-command

The macrogravity field in the centrifuge basket precludes any human intervention. All on-board equipment is remotely guided from the control room.

The movement applied to the Rolling box is controlled by a servo-controller manufactured by MOOG. A control-loop was created with the rotation sensor which is an absolute single turn encoder with a precision of 21 bits. The software associated with the controller allows a real time control of the movement applied to the Rolling box. The movement could be a sine signal, or other signal as required.

Instrumentation

The Data acquisition System HBM Spider enables conditioning and digitizing measurements by means of synchronous 8-channel modules that may be linked. Any type of sensor may be conditioned: full bridge, half-bridge, voltage source, temperature probe… The sampling frequency reaches 1.2 kHz.
Pore pressure measurements are necessary to evaluate the overpressure generated by the mechanical solicitations and to compare to the effective stress for liquefaction analysis. The sensors used classically are Druck or Measurement sensors with a range of 700 kPa.

Earth pressure sensors Kyowa (200 and 500 kPa) will be installed on the walls of the box and at the bottom.

A digital camera will be installed in front of the glass of the container. This one will turn with the rolling box to observe the movement of the cargo. This full HD colour camera allows observation and measurement of the phenomenon. If measurement needs more precision, a higher definition camera will be installed.

Small size B&K IEPE accelerometers could be installed in the cargo.

Roll angle can be measure with the rotation sensor installed to control the Rolling box.

Pressures sensors are installed on the hydraulic inputs of the actuator to verify the approximate torque issued by the system.

**Performances**

The performances have been selected to simulate one degree of freedom of a rigid vessel rolling movement during shipment. Thanks to the centrifuge technique, the stresses and pressures are similar to the ones encountered in the vessel. The technical characteristics are presented in Error! Source du renvoi introuvable.

<table>
<thead>
<tr>
<th>Max. g-level</th>
<th>Maximum angular velocity [°/s]</th>
<th>Maximum rotation angle of Rolling [°]</th>
<th>Mass of the box (empty) [kg]</th>
<th>Maximum mass of material in the box [kg]</th>
<th>Maximum moving mass [kg]</th>
<th>Total mass of the device (empty) [kg]</th>
</tr>
</thead>
<tbody>
<tr>
<td>80</td>
<td>60</td>
<td>±25</td>
<td>140</td>
<td>40</td>
<td>400</td>
<td>1485*</td>
</tr>
</tbody>
</table>

*Including centrifuge container

A proof test has been performed up to 80×g. The box, filled with water, has been tested at the lowest elevation under a frequency of 0.1Hz. The movements follow a sine signal and pressures in the system were in line with the expectations.

**3 CONCLUSIONS**

A new device has been developed to simulate Rolling test for studying the liquefaction hazards of ore cargo. Designed for centrifuge testing at 80×g, it has been successfully tested under those conditions in the framework of approval testing. In the future, the first tests with ore cargo will concern iron concentrate and lateritic-nickel ore. The objectives are: 1) to observe; 2) to understand; 3) to simulate liquefaction of ore cargo and 4) to test counter-measures on small-scale models to avoid this phenomenon.
4 ACKNOWLEDGMENTS

This research has been supported by the programme ERANet MarTec (Maritime Technologies) via the French Ministry of Ecology, Sustainable Development and Energy, General Direction of Infrastructures, Transports and Sea (DGITM/SAGS/EP1). They are greatly acknowledged.

5 REFERENCES

A NEW ADJUSTMENT-FREE DAMPING METHOD FOR FREE-SURFACE WAVES IN NUMERICAL SIMULATIONS

JANEK MEYER*, KAI GRAF† AND THOMAS SLAWIG§

*Yacht Research Unit Kiel
R&D-Centre Univ. Applied Sciences Kiel
Schwentinestrasse 24, 24149 Kiel, Germany
e-mail: Janek.Meyer@yru-kiel.de, web page: http://www.yru-kiel.de

† University of Applied Sciences Kiel
Department of Mechanical Engineering
Grenzstrasse 3, 24149 Kiel, Germany
e-mail: kai.graf@fh-kiel.de - Web page: http://www.fh-kiel.de

§ Kiel University
Algorithmic Optimal Control
Christian-Albrechts-Platz 4, 24118 Kiel, Germany
e-mail: ts@informatik.uni-kiel.de - Web page: http://www.algopt.informatik.uni-kiel.de

Key words: Damping of free-surface waves, Volume-of-Fluid method, RANSE, OpenFOAM

Abstract. Simulating free-surface flow around ships in sea waves using RANSE-Methods usually requires damping of the waves in front of the outlet to avoid reflections. It has been shown, that common damping methods like sponge layer methods deliver a reliable damping for monochromatic waves, but require a parameter adjustment by the user for different wave scales. The paper describes a new wave damping method which delivers results of same accuracy (reflections less than 2%) but does not require a manual user adjustment. The method is based on damping the vertical velocity component to reduce wave propagation. This is done by implicitly relaxing this component to zero. The relaxation is implemented with the deferred correction approach. The method is implemented in our own in-house OpenFOAM solver, which is a RANSE code using the volume of fluid method and a SIMPLE-like algorithm for the solution. Verification is done in 2D for waves of different scales, steepness, computational meshes and damping zone measures. A comparison to a linear sponge layer approach is given for the different wave scales. The 2D simulations show, that the best wave damping is achieved with the same relaxation function parameters for each individual wave. A 3D application to a modern yacht in head waves is presented. All simulation results show that the new method delivers a reliable wave damping without any parameter adjustment. The method is particularly applicable for flows with waves of different scales, like sea waves superposed with the wave system generated by a yacht.
1 INTRODUCTION

Predicting the motion of and the flow around yachts in waves using RANSE-solvers is not very common for flow analysis of sailing yachts. One of the obstacles is the generation of proper waves using free-surface flow methods. Especially the outlet of the flow domain produces unwanted reflections of the waves without using proper methods. Different methods suppressing such reflections have been developed leading from satisfying to unsatisfying results.

One group of damping methods can be classified as sponge layer methods. They are based on a damping-zone next to the boundary in which a source term is added to the governing equations. The source term usually weakens the vertical component of the fluid velocity which prevents the wave of moving through this zone. This methods deliver a good result with reflections less than 2% but require a parameter adjustment by the user.

Another way using a damping zone is presented in [1]. It explicitly relaxes the velocity to zero and the volume fraction to values of an undisturbed free-surface. This method introduces some numerical problems due to the explicit manipulation of the results of the Reynolds-averaged Navier-Stokes equations.

In [2] a method suppressing reflections at the outlet based of active filtering is presented. This method manipulates the outlet boundary condition without using a damping-zone. Indeed the damping quality is not satisfying and reflection up to 15% occure.

A lot of approaches preventing wave reflections have been given, but to the authors knowledge none of them deliver a reliable quality without parameter adjustment.

Wave damping is not only interesting for ships in waves. Also the simulation of free-surface flow around ships without sea waves may benefit of a wave-damping method. Here, the waves generated by the ship are reflected and inhibits a 100% steady-state solution. Stretching the grid in front of the outlet will prevent reflections, but a proper wave-damping method might reduce the effort for the user. Furthermore simulating offshore structures in waves require an adequate method to prevent wave reflections.

2 GOVERNING EQUATIONS AND SOLUTION METHOD FOR FREE-SURFACE FLOW

For the calculation of the free-surface flow the incompressible unsteady Reynolds-averaged Navier-Stokes equations are solved using the finite volume method. The Volume-of-Fluid (VOF) method introduced in [3] is used for the calculation of the free-surface. The momentum conservation equation (employ the eddy-viscosity hypothesis for closure), the mass conversation and the conservation equation for the transport of the volume fraction $\alpha$ are defined as

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) - \nabla \cdot (\mu_e (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)) = -\nabla p + \rho \mathbf{g}$$

$$\nabla \cdot \mathbf{u} = 0$$

$$\frac{\partial \alpha_i}{\partial t} + \nabla \cdot (\alpha_i \mathbf{u}) = 0$$

with the volume fraction $\alpha_i$ for the $i$th fluid of the two phases water and air, the velocity vector $\mathbf{u}$, the pressure $p$, the gravity vector $\mathbf{g}$, the density $\rho$ and the effective dynamic viscosity $\mu_e$. The flow properties are then calculated by $\rho = \sum_i \rho_i \alpha_i$, $\mu = \sum_i \mu_i \alpha_i$ and $1 = \sum_i \alpha_i$. The free-surface
is defined by the volume fraction $\alpha = 0.5$. The linearized, semi-discretized momentum equation can be written as

$$a_d u^{q+1}_d + \sum_n a_n u^{q+1}_n = -\nabla p^{q+1} + s_{w/o \ p}.$$  \hspace{1cm} (4)

Here, $a$ represents the elements of the coefficient matrix $A$ and the subscripts $d$ and $n$ mark the main diagonal- and neighbor-elements. All sources and contributions to the right hand side except the pressure gradient are included in $s_{w/o \ p}$. The solution of the current SIMPLE-iteration is marked with $q+1$ and the solution of the last iteration with $q$. We will not distinguish between the first prediction of the velocity and the corrected velocity of the same iteration. Rearranging (4) to $u_c$ yields the velocity equation:

$$u^{q+1}_d = \frac{1}{a_d} \left( -\nabla p^{q+1} + s_{w/o \ p} - \sum_n a_n u^{q}_n \right). \hspace{1cm} (5)$$

Substituting and rearranging (5) into (2) yields the poisson equation for the pressure:

$$\nabla \cdot \left( \frac{1}{a_d} \nabla p^{q+1} \right) = \nabla \cdot \frac{1}{a_d} \left( s_{w/o \ p} - \sum_n a_n u^{q}_n \right). \hspace{1cm} (6)$$

After integrating over the volume, the Gauss Theorem is used to transform the volume-integrals to surface-integrals. Face-variables are then interpolated with second-order discretization schemes. The pressure and its gradient is reconstructed with a method for arbitrary unstructured grids given in [4]. This prevents smearing of the density induced jump behavior at the free-surface. This includes the jump in the characteristic of the surface normal gradient of the pressure at the free-surface. Discretization in time is done with OpenFOAM’s backward scheme, which is a full-implicit, second-order scheme based on quadratic interpolation using the values of two old time steps. The convective-term of the volume-of-fluid equation is discretized with the *Blended Interface Capturing Scheme with Reconstruction* (BRICS) as described in [5]. The rigid body motion is calculated with an in house motion-solver for OpenFOAM considering virtual added mass as described in [6]. This guarantees solver stability and a optimal convergence behavior even for relatively light ships like sailing-yachts. The motion-solver has been implemented and shared with us by FluidEngineeringSolutions GmbH & Co. KG, Germany. The equations are solved in a segregated algorithm and following steps are done in each time step: (A) if required solve motion eqs., (B) solve VOF-equation, (C) solve momentum predictor, (D) solve pressure eq., (E) update flux and velocity, (F) solve turbulence equations. Inside step (D) the pressure eq. may be solved a number of times for non-orthogonal correction. Steps (D) to (E) may be repeated to apply a Piso-like Correction. Steps (A) to (F) are repeated and yield the typical outer iterations of the SIMPLE-algorithm. The momentum and VOF equations are relaxed implicitly. The new solution of the pressure is relaxed explicitly before correcting the velocity. A detailed description of the solver and solution method is given in [7].

### 3 COMMON WAVE DAMPING METHODS

Two widely used reliable wave damping methods have been described in [8] and [9], whereby the first will be described here. The method is based on a sponge layer which can be derived
by the typical equations for porous media. The damping is achieved by adding a source term to the momentum equation inside a user-defined damping-zone in front of the outlet. The source term is added to the vertical z-component of the momentum equation and can be written as

\[ s^d_z = -\rho (f_1 + f_2|u_z|) w u_z \]  

with the weight-function

\[ w = \frac{e^\kappa - 1}{e^1 - 1} \]  

and the character of the blending function

\[ \kappa = \left( \frac{x - x_{sd}}{x_{ed} - x_{sd}} \right)^\zeta \]  

with \( \zeta \) usually set to 3.5. Here \( \rho \) is the density of the fluid and \( u_z \) is the vertical velocity component. The parameter \( f_1 \) gives the amount of linear damping, \( f_2 \) the amount of quadratic damping. The weight factor \( w \) depends on the location inside the domain and helps to smoothly fade in the source term in the damping zone. The wave propagation direction is given by \( x \) with \( x_{sd} \) as the start and \( x_{ed} \) as the end x-coordinate of the damping zone. This method is implemented in commercial codes like STAR-CCM+ or in a slightly different form in ANSYS Fluent. It has been deeply investigated in [10] and it has been shown, that the method delivers a reliable damping with satisfying damping quality. Nevertheless the parameters \( f_1 \) and \( f_2 \) have to be adjusted for different waves. Scaling laws for adjusting these parameters are also given in [10]. Assuming optimal chosen parameters for a regular and monochromatic wave adjustment is necessary if the wave changes in its scale. Where no adjustment is required if the computational mesh or wave steepness is changed. Also no adjustment is required for different lengths of the damping zone. However, the maximal achievable damping quality depends on this length and at least two wavelengths are recommended.

4 DERIVATION OF THE NEW WAVE DAMPING METHOD

In [1] wave damping is achieved by relaxing the velocity \( u \) and the volume fraction \( \alpha \) explicitly after solving for the volume fraction. Explicit relaxation is done with the generic equation

\[ \phi_{\text{relaxed}} = r \phi + (1 - r) \phi^t \]  

Here \( \phi \) is an generic quantity, the superscript \( t \) signifies the target value. The relaxation factor \( r \) depends on above mentioned damping weight \( w \):

\[ r = 1 - w \]  

The method delivers a good damping quality but has significant disadvantages. All three velocity components are relaxed to zero. The volume fraction is relaxed to values which assume an undisturbed flat free-surface at constant height. This forbids additional current or boat speed superposing with the orbital velocity of the waves. Additionally this delivers some kind of Dirichlet boundary condition (BC) at the outlets, whereby a Neumann BC is desirable. Furthermore
such an explicit relaxation will prevent the convergence of the SIMPLE-algorithm. Still, this relaxation approach and the sponge layer approach described in section 3 are the inspiration for our method. Our goal is to reduce the vertical velocity component by the use of an implicit relaxation included in the momentum equation. Considering the discretized momentum equation \( \mathbf{A} \cdot \mathbf{u} = \mathbf{s} \) with the coefficient matrix \( \mathbf{A} \), the velocity vector \( \mathbf{u} \) and all explicitly treated terms on the right hand side included in the source \( \mathbf{s} \), implicit relaxation of all velocity components can be done with

\[
\frac{1}{r} a_d u_{q+1}^d + \sum_n a_n u_{q+1}^n = s_z + \frac{1 - r}{r} a_d u_{d}^1 .
\]  

(12)

Here, the subscripts \( d \) marks the main diagonal- and \( n \) the neighbor-elements of \( \mathbf{A} \). As the limiting case, where the relaxation factor \( r \) tends to zero the convergence of the equation system is not obvious. Adopting L’Hôpital’s rule twice one can show that \( u_{q+1}^d \) tends to \( u_d^1 \) as desired.

Modifying equation (12) to relax only the vertical velocity component is not straightforward. Using this approach the coefficient matrix \( \mathbf{A} \) has to be modified. Indeed it is common practice to reuse this matrix for all three components. Therefore manipulating \( \mathbf{A} \) only for the vertical component will produce additional calculation effort and a lot of programming effort to implement this methods into existing numerical codes like OpenFOAM. To solve this problem our idea is to implement the implicit relaxation with the help of the deferred correction approach. That means, the product of the modified matrix and the velocity is treated explicitly on the right hand side. Additionally the product of the unmodified matrix and the velocity is added on both, explicit and implicit, sides leading to the original unmodified left hand side. If the equation system is converging, the terms with the unmodified matrix are canceling each other out and the solution depends only on the modified matrix. This allows to modify only the right hand side, more precisely only the \( z \)-component of the right hand side. In the following we will describe two approaches to use the deferred correction. The first approach might be the obvious way to go but leads to a diverging equation system, as it will be shown. The second approach leads to our final damping method and a converging equation system.

4.1 First Approach (divergent)

In the following equations all terms on the left hand side consider the unknown velocity from the current iteration \( q+1 \) and all terms on the right hand side use the known values of the last iteration \( q \). For convergence the velocity \( u^q \) should tend to \( u^{q+1} \). Starting from the \( z \)-component of the vector equation (12)

\[
\frac{1}{r} a_d u_{q+1}^zd + \sum_n a_n u_{q+1}^zn = s_z + \frac{1 - r}{r} a_d u_{d}^1 .
\]  

(13)

and applying \( a_d u_{zd} \) on both sides leads to

\[
\frac{1}{r} a_d u_{q+1}^zd + a_d u_{q+1}^zd + \sum_n a_n u_{q+1}^zn = s_z + \frac{1 - r}{r} a_d u_{zd}^1 + a_d u_{zd}^q .
\]  

(14)

Putting the term \( \frac{1}{r} a_d u_{zd}^q \) from the left to the right hand side changes its iteration index and leads to

\[
a_d u_{zd}^{q+1} + \sum_n a_n u_{zd}^{q+1} = s_z + \frac{1 - r}{r} a_d u_{zd}^1 + a_d u_{zd}^q - \frac{1}{r} a_d u_{zd}^q .
\]  

(15)
Simplifying this equation delivers the final equation for the first approach

\[ a_d u_{zd}^{q+1} + \sum_n a_n u_{zn}^{q+1} = s_z + \frac{1-r}{r} \left( a_d u_{zd}^l - a_d u_{zd}^q \right). \]  \hspace{1cm} (16)

The whole relaxation of the z-component for the velocity is included in one source term on the right hand side. Therefore it is no more necessary to manipulate the coefficient matrix \(A\) and it is possible to include the relaxation of the z-component in the typical vector form of the momentum equation used in codes like OpenFOAM.

### 4.2 Investigation on the convergence behavior - 1st approach

Using the Jacobi method to solve equation (16) leads to

\[ u_{zd}^{q+1} = \frac{1}{a_d} \left( s_z + \frac{1-r}{r} \left( a_d u_{zd}^l - a_d u_{zd}^q \right) - \sum_n a_n u_{zn}^q \right). \]  \hspace{1cm} (17)

For wave damping \((u_{zd}^l = 0)\) with full relaxation \((r \to 0)\) the source term \(s_z\) becomes negletable and the solution of the equation is \(u_{zd}^{q+1} \to -\infty u_{zd}^q\). Therefore no convergence is possible for small \(r\).

### 4.3 Second Approach

For the second approach equation (13) is multiplied with the relaxation factor \(r\) before applying the deferred correction

\[ a_d u_{zd}^{q+1} + r \sum_n a_n u_{zn}^{q+1} = r s_z + (1-r) a_d u_{zd}^l. \]  \hspace{1cm} (18)

In eq. (13) of the first approach the main diagonal elements of \(A\) are multiplied with \(\frac{1}{r}\). Now, in eq. (18) the neighbor elements of \(A\) and the right hand side term \(s_z\) are multiplied with \(r\). Applying the deferred correction method to eq. (18) to get rid of the relaxation factor on the left hand side leads to

\[ a_d u_{zd}^{q+1} + \sum_n a_n u_{zn}^{q+1} = r s_z + (1-r) \left( a_d u_{zd}^l + \sum_n a_n u_{zn}^q \right). \]  \hspace{1cm} (19)

Here, the original right hand side term \(s_z\) is manipulated. This needs to be considered in the derivation of the pressure equation. Our solution how to consider this as easy as possible is described in subsection 4.5.

### 4.4 Investigation on the convergence behavior - 2nd approach

Using the Jacobi method again to solve eq. (19) gives

\[ u_{zd}^{q+1} = \frac{1}{a_d} \left( r s_z + (1-r) \left( a_d u_{zd}^l + \sum_n a_n u_{zn}^q \right) - \sum_n a_n u_{zn}^q \right). \]  \hspace{1cm} (20)

For full relaxation \((r \to 0)\) one gets the aspired behavior \(u_{zd}^q \to u_{zd}^l\) where \(u_{zd}^l\) is zero for wave damping.
4.5 Manipulation of the original right hand side

Using equation (19) requires to manipulate the original right hand side $s_z$ including the terms depending on the pressure $p$. Therefore it is necessary to build a new pressure equation. This can be done straightforward but is not our finally chosen way. For the sake of completeness this way will be given here, first. Afterward a simpler way leading to our final method will be described.

First way:

The vector form of the momentum equation considering the wave damping method can be written as:

$$a_d u_{d}^{q+1} + \sum_n a_n u_{n}^{q+1} = R \cdot (-\nabla p^q + s_{w/o \ p}) + s^* .$$

(21)

Here, $s_{w/o \ p}$ is the original right hand side without the pressure gradient ($s = -\nabla p + s_{w/o \ p}$).

The tensor $R$ allows to manipulate only the z-component of the equation and is defined as $R = \begin{bmatrix} r_x & 0 & 0 \\ 0 & r_y & 0 \\ 0 & 0 & r_z \end{bmatrix}$ with the relaxation factors $r_x = r_y = 1.0$ and $r_z = r$ for the three Cartesian directions. The source term $s^*$ is defined as

$$s^* = (\delta_{ij} - R) \cdot \left( a_d u_{d}^{q} + \sum_n a_n u_{n}^{q} \right) .$$

(22)

with the Kronecker-delta $\delta_{ij} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$. Rearranging eq. (21) for $u_{d}^{q+1}$ leads to the new velocity equation allowing to calculate the corrected velocity $u_{d}^{q+1}$

$$u_{d}^{q+1} = \frac{1}{a_d} \left( R \cdot (-\nabla p^{q+1} + s_{w/o \ p}) + s^* - \sum_n a_n u_{n}^{q} \right) .$$

(23)

Substituting the velocity of the continuity equation (2) with (23) and rearranging gives the adopted pressure equation

$$\nabla \cdot \left( \frac{1}{a_d} R \cdot \nabla p^{q+1} \right) = \nabla \cdot \frac{1}{a_d} \left( R \cdot s_{w/o \ p} + s^* - \sum_n a_n u_{n}^{q} \right) .$$

(24)

Second way:

For the second way the momentum equation is written in a way containing the unmodified right hand side $s$ and an additional source term

$$a_d u_{d}^{q+1} + \sum_n a_n u_{n}^{q+1} = -\nabla p^q + s_{w/o \ p} + s_{\text{wavedamping}}$$

(25)

with

$$s_{\text{wavedamping}} = (\delta_{ij} - R) \cdot \left( \nabla \tilde{p}^q - s_{w/o \ p} + a_d u_{d} + \sum_n a_n u_{n} \right) .$$

(26)
It is important to notice, that the pressure gradient is used in two terms. To build the pressure equation it is only solved for the pressure of the original right hand side \( s \). The pressure gradient inside the source term for the wave damping \( s_{\text{wavedamping}} \) is fixed and therefore marked with a tilde. The equation for the corrected velocity becomes

\[
\mathbf{u}_{d}^{q+1} = \frac{1}{a_{d}} \left( -\nabla p^{q+1} + s_{w/o \ p} + s_{\text{wavedamping}} - \sum_{n} a_{n} \mathbf{u}_{n}^{q} \right) \tag{27}
\]

and the pressure equation becomes

\[
\nabla \cdot \left( \frac{1}{a_{d}} \nabla p^{q+1} \right) = \nabla \cdot \left( \frac{1}{a_{d}} \left( s_{w/o \ p} + s_{\text{wavedamping}} - \sum_{n} a_{n} \mathbf{u}_{n}^{q} \right) \right). \tag{28}
\]

Because \( s_{\text{wavedamping}} \) contains the pressure of the last iteration \( p^{q} \) the solution of the pressure could be interpreted as deferred inside the wave damping zone.

This is our final wave damping method made up of only one additional source term. The original right hand side \( s \) can be pre-calculated and then reused to build the wave-damping source term \( s_{\text{wavedamping}} \). The weight function \( w \) included in the relaxation factor \( r \) has not the final form and will be substituted with the optimized function \( w_{\text{opt}} \) as described in chapter 6.3.

5 WAVE GENERATION

At the inlet the values for the velocities and volume fractions are prescribed according to the wave theory. An equation for the pressure value itself or the pressure gradient is missing, which is a significant problem. Our solution is using a Neumann boundary condition with a zero gradient for the pressure. Certainly, the assumption of a zero gradient is wrong in the presence of a wave and for many 3D simulations the wave will collapse behind the inlet. Therefore we are using an relaxation zone behind the inlet and apply full implicit relaxation for the momentum and volume-of-fluid equation. The relaxation factor has the reversed characteristic of the relaxation factor of the damping-zone. Therefore full relaxation is applied at the inlet going to no relaxation at the end of the generation-zone. The target values \( \mathbf{u}^{t} \) and \( \alpha^{t} \) are calculate with the chosen wave theory for example 5th-order Stokes. All three velocity components are relaxed to the target value by using equation (12). The relaxation for the volume fraction is applied in the same way

\[
\frac{1}{r} a_{\text{VOF} \ d} \alpha_{d}^{q+1} + \sum_{n} a_{\text{VOF} \ n} \alpha_{n}^{q+1} = s_{\text{VOF}} + \frac{1 - r}{r} a_{\text{VOF} \ d} \alpha_{d}^{t} \tag{29}
\]

with the diagonal elements \( a_{\text{VOF} \ d} \) and neighbor elements \( a_{\text{VOF} \ n} \) of the coefficient matrix of the volume-of-fluid transport equation. The term \( s_{\text{VOF}} \) contains all possible source terms or contributions to the right hand side.

We have not systematically tested different zone length, but a length of 1.0\( \lambda \) produces a reliable wave generation for all of our simulations.

The free-surface is initialized wave-less in the hole domain. For the wave generation we are using the T-soft-parameter of the waves2Foam library. This generates a wave growing from a smooth surface to the full height over the given time \( T_{\text{soft}} \) in the generation-zone. The growing time \( T_{\text{soft}} \) is set to the wave-period \( T \). Due to the relaxation approach, the relaxation factor
becomes small at the end of the generation zone. Therefore, while growing the wave gets smeared at the end of the zone, because the values are set to the new height only partially which leads to very small oscillations in the wave. Still this is acceptable and our best method to produce waves. The oscillations are quantified in chapter 6.

6 VERIFICATION AND WEIGHT FUNCTION OPTIMIZATION

To estimate the damping quality of our method a test-case very close to the one presented in [10] has been chosen. The test-case is also used to optimize the function for the relaxation factor for monochromatic waves. Additionally we have implement the typical sponge layer method for comparison.

6.1 Numerical Simulation Setup

The test-case is based on a 2D wave tank. Figure 1 shows the domain and its measurements. The wave is generated in the generation zone with a length of 1\(\lambda\). Afterward the wave is propagated through a region of length 4.0\(\lambda\). Finally the wave damping is applied in the damping zone of length 2.0\(\lambda\). The Reflections are measured inside the measurement zone in front of the damping zone as explained later. The domain has a height of 4.5\(\lambda\) filled with water up to 4.0\(\lambda\). The grid has a resolution of 20 cells per wave-height and 100 cells per wave-length as recommended in [10] which refers to [11]. The grid is coarsened with growing distance to the wave. The waves are generated by prescribing volume fractions and velocities of 5th-order Stokes waves at the inlet boundary and in the generation zone as explained in chapter 5. A Neumann boundary condition (BC) with zero gradient is applied for the pressure at the inlet. A free slip wall is applied at the bottom. Indeed, instead of using the hydrostatic pressure gradient a Dirichlet BC with given hydrostatic pressure is applied for the pressure at the bottom. This is done to prevent problems with the preservation of the position of the free-surface. Same BC is applied at the top with the exception, that the velocity BC is set to a zero gradient BC to get a behavior as an open water tank. At the outlet a zero gradient BC is applied for all variables. This allows a lowered free-surface at the outlet or a wave traveling through the outlet.

The solver is using a time step of about \(\frac{1}{500}\) of the wave period and 10 SIMPLE-Loops. The pressure is relaxed by 0.3, the velocity by 0.7 and the volume fraction by 0.9. The simulation is assumed to be laminar and no turbulence-model is applied.

Simulations have been done for different wave scales; a medium sized wave with \(\lambda = 4.0m\) and \(H = 0.16m\), a small wave with \(\lambda = 0.04m\) and \(H = 0.0016m\) and a big wave with \(\lambda = 400m\) and \(H = 16.0m\). To compare the dependency of the wave steepness a steep wave with \(\lambda = 0.4\) and \(H = 0.16\) has been simulated. The grid has been scaled accordingly.
The influence of the grid is investigated by using a coarse grid with 10 cells per wave height and 50 cells per wave length for the medium sized wave. The influence of the damping zone length is investigated by also using a damping zone with a length of $1\lambda$ for the medium sized wave. The grid is the same as for the initial, medium sized wave simulation. The measurement zone is also kept at the initial position.

6.2 Quantifying the damping quality

To quantify the damping quality, the maximal and minimal wave heights $H_{\text{max}}$ and $H_{\text{min}}$ are measured in the measurement zone in the time interval $[22.0T, 24.0T]$ at 40 evenly distributed times. These wave heights are used to build a reflection coefficient as suggested in [12]

$$C_{\text{R}} = \frac{H_{\text{max}} - H_{\text{min}}}{H_{\text{max}} + H_{\text{min}}} \quad (30)$$

For perfect wave reflection of 100% the coefficient $C_{\text{R}}$ becomes 1.0 and zero for no reflection. Considering that the wave train moves with the half speed of a single wave, the chosen time interval allows, that the wave train propagates from the inlet to the outlet and after reflection back to the measurement zone. The wave height is estimated by measuring the minimal and maximal z-Position of the free surface inside the measurement zone. Please notice, that the reflection coefficient includes the wave reflections, but also wave oscillations originating from the wave generation. Additionally a wave, flattening inside the measurement zone due to an insufficient discretization of time or of the convective term will influence the reflection coefficient. The superposition of all these phenomena may influence the coefficient negatively, but also positively. Reflections may occur at the outlet but also inside or at the beginning of the damping zone, due to too much dampening. To estimate the background oscillations (BO) originating from the wave generation and the influence of flattening all simulations have been done without wave damping but with a long domain of length $25.0\lambda$.

6.3 Optimizing the weight function

The weight function given in equation (8) does not necessarily guarantee the least reflections. Looking at equation (7) of the sponge layer method one can interpret the parameters $f_1$ and $f_2$ as a scaling of the weight function $w$ for the different waves. To allow an optimization for our method we are also scaling the weight function with the new introduced scale factor $\chi$

$$w_{\text{opt}} = \chi w \quad (31)$$

The scale factor $\chi$ is varied from 0.008 to 1.0 for each wave to achieve the least reflections and to show the dependency of our wave damping method from the wave characteristics.

6.4 Results

Figure 2 shows the results of all simulations. Sub-figure b shows the results of the common linear sponge layer method (with $f_2 = 0.0 = \text{const}$), also implemented in our solver. The results are remarkable close to the results presented in [10] using the same method but implemented in StarCCM+. The black, dashed line shows the limit of $C_{\text{R \; lim}} = 0.02$ for an acceptable
Figure 2: Reflection coefficient for different wave damping method and different waves
small wave: $\lambda = 0.04m$, $H = 0.0016m$; medium wave: $\lambda = 4.0m$, $H = 0.16m$; big wave: $\lambda = 400m$, $H = 16.0m$; steep wave: $\lambda = 0.4m$, $H = 0.16m$
reflection. Here, the sponge layer method has only been used for different wave scales and one can clearly see that \(f_1\) has to be adjusted to achieve an acceptable reflection. For the quadratic sponge layer method we would like to refer to the results of [10], showing that \(f_2\) has to be adjusted, too. Sub-figure c shows the results of our new method with both axis scaled logarithmically. The initial background oscillations (BO) are marked with a dashed line in the left area of the diagram. All background oscillations are below the maximal allowed value. This shows that the oscillations produced by the wave generation method mentioned in chapter 5 are acceptable. All curves show the same characteristics and an optimal wave damping is achieved for approximately \(0.03 \leq \chi \leq 0.15\). The absolute minimum of each curve corresponds with the background oscillations. Only the simulation with the short damping zone clearly shows a worse damping quality. Additionally the absolute minimum for the steep wave deceeds the corresponding background oscillations. A clear reason can not be given but it is possible that the background oscillations and the reflections are canceling each other out.

The characteristics of the curves can be interpreted in that way, that the increasing reflections in the area \(\chi < 0.03\) are reflections which arises predominantly at the outlet due to too little damping and that the increasing reflections in the area \(\chi > 0.15\) are reflections which arises predominantly at the beginning or inside the wave damping zone due to too fast wave damping.

Sub-figure d shows the same results as c but with a linear scale. This should emphasize graphically that all curves have the same characteristics. Therefore our wave damping method allows to use the same weight function for all different monochromatic waves and no user adjustment is required. To make short damping zones practical we recommend a scale factor \(\chi\) of about 0.09. Nevertheless we recommend a damping zone with a length of \(2.0 \lambda_{\text{max}}\) where \(\lambda_{\text{max}}\) should be the largest wave length of a given wave spectrum. One can assume that the evaluation of the damping quality would turn out much better without the presence of the background oscillations.
Figures 3 to 5 show the wave shapes for different damping parameters. The end of the generation-zone (left) and the beginning of the damping-zone (right) are marked with vertical red lines. Figure 3 shows the wave after 40s without any wave-damping applied, leading to reflections of 42%. One can clearly see the reflections superposing with the original wave. Figure 4 shows the result arising due to too much damping, which still leads to only 1.6% reflection. Here, the free-surface becomes completely flat in front of the outlet. Figure 5 shows the wave shape arising due to optimal wave-damping leading to only 1.1% reflection. Here, still a very small wave reaches the outlet. Compared to 4, this will even more reduce reflections at the beginning of the damping-zone. Furthermore, possible reflections from the outlet will be damped while traveling back through the damping-zone.

7 APPLICATION TO A YACHT IN HEAD WAVES

This section shows an application of the new wave damping method to a yacht in head waves. This is a good example for the superposition of the sea waves with the small waves of the yacht and furthermore with the velocity of the yacht.

The yacht is a Class 40 (C40) designed by VPLP, France, and has a length over surface of $\text{LoS} = 12\text{m}$. The waves have a length of $\lambda = 18\text{m}$, a height of $H = 0.4\text{m}$ and a period of $T = 3.489\text{s}$. Second order Stokes theory is used for wave generation. The yacht is accelerated to $u = 4.4\text{m/s}$ in the first second. This leads to an encounter frequency of $0.53\frac{1}{\text{s}}$. The yacht is free in pitch and heave.

The solution-domain uses a wave-generation zone of $1\lambda$ and a wave-damping zone of $2\lambda$. The free flow begins $1\text{LoS}$ in front of the yacht and ends $2\text{LoS}$ behind the yacht. The grid is generated using the OpenFOAM mesher snappyHexMesh, but with the use of some small in-house code and a lot of scripting to get a mesh of good quality with anisotropic refinement. The grid uses 100 cells per wave length and 20 cells per wave height. Depending on the part of the yacht, seven to nine prism-layers are applied leading to $y+$ values of 40 to 90 underwater. The kelvin-refinement ends in front of the wave-damping zone. The cells inside the kelvin-refinement have a size less or equal to $0.125\text{m}$. The final mesh has $8.5\times10^6$ cells.

A modified $k-\omega$-SST turbulence model is used. The modification consist of a different production term as described in [1] and a correct consideration of the density-derivations for free-surface flows, compared to OpenFOAM’s standard turbulence models. The timestep was

![Figure 6: C40-design in head waves - pressure forces](image-url)
Figures 6 and 7 show the pressure and the viscous forces. One can clearly see a periodic behavior and no significant disturbance due to wave reflections. The viscous force in x-direction has the most variation. This can be explained with ventilation under the hull composed of correct ventilation due to encountering a trough and incorrect numerical ventilation due to smearing of the free-surface. First simulations using a coarser grid (4.4E6 cells, three prism layers, coarser kelvin refinement) show much more variations in the viscous forces in the x-direction.

Figures 8 and 9 show a close view onto the hull and its wave system for two different times. Figure 8 shows the hull diving into the sea wave, whereas Figure 9 shows the hull with the bow knuckle significantly above the free-surface. In both cases, one can clearly see a smooth and good resolved kelvin wave pattern and furthermore breaking waves at the hull.

Figures 10 and 11 show wave patterns at different times. The end of the generation-zone and the beginning of the damping-zone are marked with two red y-planes in Figure 11. As one can see, the waves get damped in front of the outlet. The large scaled sea waves are not dampened completely at the outlet as expected for the chosen $\chi$ of 0.09. Apart from that, the small scaled wave-system of the yacht seems to be dampened completely at the outlet. The reason for this is first the length of the damping-zone, which is significantly longer than twice the wave length of the yacht induced waves. Therefore the damping for this small-scaled waves should be of a quality better than required. The second reason is the kelvin refinement ending in front of the damping-zone. At the y-max and y-min domain borders (the sides of the domain) the free traveling waves show no significant differences to each others. This underscores first the good wave propagation due to the 2nd order time discretization and second the suppression of wave reflections.

8 CONCLUSION

A new wave damping method was derived and investigated for monochromatic waves. The simulations show an overall good damping quality with reflections less than 2%. For different wave characteristics the same parameters leads to the best damping quality. Therefore the method can be seen as adjustment-free in the scope of the investigated waves. The damping quality for irregular or breaking waves has not been investigated and is an open topic for future investigations. Nevertheless, especially for irregular waves, which can be seen as a superposition...
Figure 8: C40-design close-up at $t = 23.1$

Figure 9: C40-design close-up at $t = 23.9$

Figure 10: C40-design wave pattern at $t = 23.9$

Figure 11: C40-design 3D view at $t = 23.1$
of regular waves, the authors expect a good damping quality, due to the adjustment-free behavior for monochromatic regular waves. The application of the method to a yacht in head waves emphasizes the user-friendly applicability, as it delivers a periodic solution with less variation just by activating the damping method with our optimized parameters.

REFERENCES


Computational techniques used for velocity prediction of wing-sailed hydrofoiling catamarans

MARINE 2017

RICHARD GJ FLAY*, NILS HAGEMEISTER†

* Yacht Research Unit
University of Auckland
Private Bag 92019, Auckland 1142, New Zealand
E-mail: r.flay@auckland.ac.nz, www.yru.auckland.ac.nz

† Formerly Yacht Research Unit Auckland
University of Auckland
Private Bag 92019, Auckland 1142, New Zealand
E-mail: nhag107@aucklanduni.ac.nz

Key words: Multihull, Catamaran, Velocity Prediction, VPP, Foiling, Wing-sails

1 INTRODUCTION

Velocity Prediction Programs (VPP) are among the most commonly used tools in racing yacht design. Whether looking at a parameter-based variant with few degrees of freedom or a full 6 DOF experimental/CFD data postprocessor, their output is often critical to design decisions. In the past, research in this area has mostly concentrated on monohulls, since most major regattas were sailed in these boats.

Following the 34th America’s Cup, (foiling) multihulls have moved into the focus of elite sailors and designers. However, publically available information on velocity prediction for these boats remains relatively sparse, especially if resources do not allow for extensive wind tunnel or RANS-CFD studies. Hence the computational techniques used in this study are deliberately restricted to freely available software and self-programmed code that can be run on a standard desktop computer.

2 FORCE DECOMPOSITION

VPPs are essentially optimisation tools for the determination of trim settings for the objective of maximum speed under the constraint of the equilibrium of forces and moments acting on the yacht, as expressed in Eq. (1).

\[ \sum F = \sum M = 0 \]  

To apply the constraints, these forces have to be modelled, for which they are decomposed according to the fluid they operate in and the part of the boat they apply to.

Figure 1 shows the forces acting on the catamaran as considered in this study. The model assumes linearity of forces and small heel angles, since the righting moment is at its maximum with the windward hull just clearing the water or during level foiling. As a result vertical
aerodynamic forces can be omitted and the vertical position of the centre of gravity becomes irrelevant.

Aerodynamic forces are divided into wing and windage forces. Hydrodynamic forces are split into the contributions from the hull, daggerboard and rudder. For these the inclusion of the vertical forces is particularly important to simulate the foiling states. Dynamic lift on the hull is neglected since it is assumed to be small compared to static buoyancy and the forces exerted by the appendages plus the small error only applies to non-foiling states.

![Figure 1: Mechanic model of the catamaran](image_url)

### 3 WINDAGE

The windage force is calculated as the product of force coefficient, dynamic pressure and area, as shown in Eq. (2). This universal approach is adapted for each of the three groups into which individual parts are divided depending on their geometrical characteristics.

\[
F_i(z_i) = c_{fl} \frac{\rho}{2} \cdot AW_S(z_i) \cdot A_i
\]  

(2)

Group one contains cables and parts with circular cross-sections and small diameters in relation to their length. For these items two dimensional, laminar flow can be assumed [1], giving a force coefficient of 1.1. Changes in apparent wind speed with height are incorporated through vertical segmentation and calculation of projected areas.

The second group consists of so-called platform components, whose main extents lie in a plane approximately parallel to the water surface. Hence hulls, beams, crew and the bowsprit are believed to be affected by constant apparent wind. The treatment of these parts is inspired by a method commonly used for monohulls [1], which was modified to account for the characteristics of the catamaran. Forces are considered to act on projected frontal and lateral areas independent of the wind angle. For the lateral projected area, which changes with the wind
angle, a conservative estimation of twice the area of a single hull has been used, as it is not clear how much flow reestablishment the separation distance between the hulls allows. A force coefficient of 0.6 was used for group two as suggested by FOSSATI [1]. Furthermore the lateral windage force is taken into account in this study since it influences the side force that has to be produced by the appendages.

Group three contains mostly smaller pieces of equipment and other items with unknown aerodynamic characteristics. Their contribution is estimated at ten percent of platform windage.

4 LIFTING LINE METHODS

4.1 Wing Forces

To capture the effects of flow around wings of finite span and the variation of apparent wind with height as well as the planform and trim of the wing, the forces acting on the wing are calculated by means of a lifting line method as described by GRAF et al. [2], which is summarised in the following.

Lifting Line methods are the expansion of the Kutta-Joukowski-theorem into the third dimension. The lifting vortex is elongated normal to the aerofoil section, resulting in a vortex filament. At the wing tips this vortex filament is aligned with the incident flow and extended to infinity to satisfy Thompson’s law. The result is a so-called horseshoe vortex consisting of the bound vortex and two free vortex filaments. Changes in lift are modelled by altering vorticity through shedding of a free vortex filament [3]. A vortex sheet is created by decreasing the distance between two neighbouring free vortex filaments to zero. The influence of shed vorticity is expressed as velocity induced normal to incident flow and span, which reduces the effective angle of attack. The approach presented below assumes a bound vortex parallel to the vertical axis which simplifies the mathematics, but restricts the longitudinal alignment of the wing sections and the rake of the wing.

Figure 2 illustrates the discretisation of the wing. A horseshoe vortex consisting of the bound vortex along the quarter-chord line and the root and tip vortices represents the basis. Vortex sheets are distributed at constant intervals along the bound vortex, forming panels limited by
their upper and lower borders. The induced velocity is calculated at collocation points positioned in each panel’s vertical centre.

Figure 2: Wing discretisation

Biot-Savart’s law is applied to determine the influence of a free vortex (orange highlighting) filament in collocation point \( P_c \) [4], Eq. (3).

\[
v_{indc} = -\frac{1}{4\pi \Delta z} \Gamma
\]

(3)

The effect of a vortex sheet created by the linear change in vorticity \( (\Gamma_2 - \Gamma_1) \) between \( z_1 \) and \( z_2 \) is computed by Eq. (4) [2].

\[
v_{indc} = -\frac{1}{4\pi} (\Gamma_2 - \Gamma_1) \ln \left( \frac{z_2 - z_c}{z_1 - z_c} \right)
\]

(4)

The total velocity induced at \( P_c \) is calculated as the sum of the induced velocities of all panels plus the contributions of the root and tip vortices. Self-induction is excluded since the natural logarithm is not defined for a negative argument. To model the weakening of the root vortex through the endplate effects of the platform, the factor EP is adopted [2], Eq. (5).

\[
v_{indc} = \left[ \Gamma_0 \right]_j - \frac{1}{4\pi} \left[ \Gamma_i - \Gamma_{i-1} \right] \ln \left( \frac{z_i - z_c}{z_{i-1} - z_c} \right) - EP \left[ \Gamma_0 - \Gamma_{f_{root}} \right]_j + \frac{1}{4\pi} \left[ \Gamma_N \right]_j - \left[ \Gamma_{f_{tip}} \right]_j
\]

(5)

Eq. (5) can be rewritten by summarising the geometric relations into factors, Eq. (6).

\[
\left[ \Gamma_0 \right]_j - \frac{1}{4\pi} \left[ \Gamma_i - \Gamma_{i-1} * f_{sheet} \right]_j - EP \left[ \Gamma_0 - \Gamma_{f_{root}} \right]_j + \frac{1}{4\pi} \left[ \Gamma_N \right]_j - \left[ \Gamma_{f_{tip}} \right]_j
\]

(6)
Since the induced velocities have to be known at the boundaries of each centre to determine the vorticities for the next iteration, induced velocities are interpolated from two neighbouring panels. At the root and tip of the wing they are extrapolated [2], as shown in Eqs. (7-9).

\[ v_{ind} = \frac{(v_{ind\ell+1} + v_{ind\ell})}{2} \quad (7) \]
\[ v_{ind0} = 2v_{ind\ell} - v_{ind1} \quad (8) \]
\[ v_{indN} = 2v_{ind_{N-1}} - v_{indN-1} \quad (9) \]

Vorticity is derived from Kutta’s law which specifies lift as the product of density, velocity and vorticity and the equivalent definition of lift as function of lift coefficient, dynamic pressure and reference area [2], Eq. (10).

\[ L = \rho_{air} \cdot u \cdot \Gamma = \frac{\rho}{2} \cdot u^2 \cdot A \cdot c_L \quad (10) \]

The lift coefficient depends on the Reynolds number and the angle of attack, which is affected by the induced velocities [2], as shown in Eq. (11).

\[ AoA_{eff} = AoA - \frac{v_{ind}}{u} \quad (11) \]

Combining Eqs. (10) and (11) results in Eq. (12) [2].

\[ \Gamma = \frac{u}{2} \cdot A \cdot c_L \left( AoA - \frac{v_{ind}}{u}, Re \right) \quad (12) \]

Induced drag is proportional to the ratio of induced to incident velocity and lift. Parasitic profile drag is added to obtain total drag [2]. The required drag coefficient is obtained by interpolating in a table listing it as function of angle of attack and the Reynolds number, Eq. (13).

\[ D_{total} = \frac{v_{ind}}{u} \cdot L + \frac{\rho_{air}}{2} \cdot u^2 \cdot A \cdot c_{dp}(AoA_{eff},Re) \quad (13) \]

To enable trimming of the wing two parameters have been introduced. The angle between the root section and centreline of the yacht is denoted as \( \alpha_{CL} \) whereas twist is defined as the angle differential between root and tip section. Twist is varied linearly along the span, Eq. (14).

\[ AoA(z) = AWA(z) - \alpha_{CL} \cdot \frac{\text{twist}}{\text{span}} \cdot (z - z_0) \quad (14) \]

Since the wing operates within the part of the planetary boundary layer with the highest velocity gradients, its vertical position can be altered according to the elevation of the platform.

4.2 Appendage forces

The forces produced by the appendages are calculated with a lifting line approach similar to that presented in Section 4.1. However, since the appendages are non-planar wings, a three-dimensional approach has been selected. Additionally, the boundary conditions have been revisited to incorporate effects of surface proximity and surface piercing when foiling.

In contrast to the vortex sheets of wing, the appendages are modelled by horseshoe vortices distributed along the span. This choice has been made to simplify the mathematical
relationships in the third dimension. The reduction in accuracy is compensated for by an increase in the number of panels. Figure 3 shows the discretisation of a three-dimensional wing.

![Figure 3: Discretization of appendages wing](image)

Induced velocities are calculated from the three dimensional form of the Biot-Savart-law [4], Eq. (15).

\[
\vec{v}_{\text{ind}} = \frac{\Gamma}{4\pi} \int \frac{\hat{r} \times d\vec{s}}{|\vec{r}|^3}
\]  

(15)

Using the definitions from figure 3 Eq. (15) can be rewritten [4] as Eq. (16).

\[
\vec{v}_{\text{ind}} = \frac{\Gamma_{n-1}}{4\pi} \frac{(\vec{r}_1 \times \vec{r}_2)}{|\vec{r}_1 \times \vec{r}_2|^2} \times \hat{r}_0 \times \left( \frac{\vec{r}_1}{|\vec{r}_1|} - \frac{\vec{r}_2}{|\vec{r}_2|} \right)
\]

(16)

The expression is transformed into Eq. (17) to avoid the singularity that occurs when vectors \(\vec{r}_1\) and \(\vec{r}_2\) are parallel [5].

\[
\vec{v}_{\text{ind}} = \frac{\Gamma_{n-1}}{4\pi} \frac{(|\vec{r}_1| + |\vec{r}_2|)(\vec{r}_1 \times \vec{r}_2)}{|\vec{r}_1||\vec{r}_2| \times ((\vec{r}_1 \times \vec{r}_2) + (\vec{r}_1 \times \vec{r}_2))}
\]

(17)

To avoid the singularity in the Biot-Savart-law for distances approaching zero and improve numerical stability, two factors \(k_i\) and \(k_{bi}\) are introduced that gradually reduce the influence of the vortices when in close proximity to each other [6]. \(e_{\text{core}}\) denotes the diameter of the viscous vortex core and was set to 0.01 m for this study. A value of 2 was selected for \(m\) in Eq. (18) based on the recommendations of ABEDI [6].

\[
k_i = \left( \frac{|\vec{r}_i \times BS|}{|BS|} \right)^m \left( e_{\text{core}}^{2m} + \left( \frac{|\vec{r}_i \times BS|}{|BS|} \right)^{2m} \right)^{-m}
\]

(18)
Using the notation from figure 3 and applying Eqs. (18) and (19) to the bound and free vortex filaments, an expression for the total influence of a horseshoe vortex can be derived [5], Eq. (20).

\[
\overrightarrow{v}_{\text{t,n-1}} = \frac{\Gamma_{n-1}}{4\pi} \left( k_2 \left( \frac{\overrightarrow{B_S}}{|\overrightarrow{r}_2|} \times \overrightarrow{r}_2 \right) + k_b \left( \frac{|\overrightarrow{r}_1| + |\overrightarrow{r}_2|}{|\overrightarrow{r}_1|} \times \overrightarrow{r}_2 \right) - k_1 \frac{\overrightarrow{B_S}}{|\overrightarrow{r}_1|} \times \overrightarrow{r}_1 \right)
\]

Again the geometrical relationships can be factored out, transforming Eq. (20) to Eq. (21).

\[
\overrightarrow{v}_{\text{t,n-1}} = \frac{\Gamma_{n-1}}{4\pi} \cdot f_{n-1}
\]

The total induced velocity at point P\(_C\) is the sum of the velocities induced by each individual horseshoe vortex, as expressed in Eq. (22).

\[
\overrightarrow{v}_i = \sum_{j=1}^{n} \overrightarrow{v}_{i,j}
\]

Local flow conditions are determined by the total induced velocity to the incident flow.

\[
\overrightarrow{w}_i = \overrightarrow{B_S} + \overrightarrow{v}_{\text{indc}}
\]

The angle of attack is derived using vector calculus, as in Eq. (24). \(\overrightarrow{q}_i\) and \(\overrightarrow{c}_i\) denote the vectors normal to the wing surface and along the chord, respectively [5].

\[
A_o A_i = \tan^{-1} \left( \frac{\overrightarrow{w}_i \times \overrightarrow{q}_i}{|\overrightarrow{w}_i| \times |\overrightarrow{q}_i|} \right)
\]

Equating the three dimensional form of the Kutta-Joukowski-theorem [4] with the well-known formula defining lift as the product of lift coefficient, dynamic pressure and area yields an expression to calculate vorticity, as shown in Eq. (25). \(\overrightarrow{s}_i\) represents the vector along the span of the wing section.

\[
\frac{\rho_{\text{water}}}{2} \cdot BS^2 \cdot A_i \cdot c_{L_i} = |\Gamma_i \cdot \rho_{\text{water}} \times \overrightarrow{w}_i | \times \overrightarrow{s}_i|
\]

Introducing local chord \(c_i\) and span \(s_i\) Eq. (25) can be transformed to Eq. (26).

\[
\frac{1}{2} \cdot BS^2 \cdot s_i \cdot c_i \cdot c_{L_i} = |\Gamma_i \cdot \overrightarrow{w}_i \times \overrightarrow{s}_i |
\]

Since vorticity has to be proportional to the lift coefficient, Eq. (26) can be transposed to result in the formula used to calculate vorticity, Eq. (27).
Richard G. J. Flay, Nils Hagemeister

\[
\Gamma_i = \frac{0.5 \times BS^2 \times s_i \times c_i \times c_{Li}}{[w_i \times s_i]} \quad (27)
\]

The total force exerted by a profile section is the sum of the parasitic profile drag and the force derived from the Kutta-Joukowski-theorem, Eq. (28).

\[
\bar{F}_i = \bar{D}_{ppt} + \rho_{water} \times \Gamma_i \times (\bar{BS} + \bar{v}_i) \quad (28)
\]

Parasitic profile drag is calculated as the product of dynamic pressure, area and drag coefficient acting along the direction of the incident flow, as given in Eq. (29).

\[
\bar{D}_{ppt} = c_{Dpp}(AoA_{eff}, Re) \times \frac{\rho_{water}}{2} \times \frac{[\bar{BS}]}{[BS]} \quad (29)
\]

The effects of surface proximity have to be considered for the appendages. Since the surface represents a constant pressure boundary, the pressure fields of the appendages deform the water surface [7]. As a result transverse waves are formed and the extents of the pressure fields are reduced. The latter is often modelled by virtual mirroring of the foil at the water surface to cancel out the foil’s influence. This approach neglects the deformation of the water surface, which can be justified by the long wavelength associated with high Froude numbers [8]. Hence the total induced velocity calculated in Eq. (22) has to be amended by the contribution of the mirror image, as shown in Eq. (30).

\[
\bar{v}_i = \sum_{j=1}^{n} \bar{v}_{i,j} + \sum_{j=1}^{n} \bar{v}_{mi,j} \quad (30)
\]

Since the densities of water and air differ by a factor of approximately 10^3, surface piercing states are modelled by setting the density of airborne sections to zero.

4.3 Implementation

Since the given equations are not independent of each other, the solution has to be found iteratively. The computation sequence given in [2] has been slightly modified from the original to reduce runtime in the present work. The sequence of computation steps is given below.

1. Assign geometric properties and apparent incident velocity to each profile section.
2. Calculate matrix of influence factors from geometric relations.
3. Determine angle of attack and derive lift coefficient for each section.
4. Obtain section vorticity from lift coefficient
5. Compute total velocity induced in each panel using geometric influence factors.
6. Check convergence criteria and repeat steps 3.-5. if necessary.
7. Calculate section lift
8. Calculate induced and parasitic drag.
9. Determine driving and side forces from sectional lift, drag and AWA and integrate forces over span.

Since the method is only valid for primarily two-dimensional flow [2], stall has to be avoided, because of separated flow having a pronounced three dimensional flow pattern. To reduce the chance of trim optimisation leading to stalled wing sections, the progression of
the lift curve was altered. Figure 4 shows an exemplary comparison between the real trend of the lift coefficient as a function of angle of attack and the altered curve used for this study. It can be seen that in reality the lift drops somewhat after the stall angle is reached. The lift then starts to increase again after a local minimum is passed. The modified curve passes the local minimum and continues its descent until zero.

![Figure 4: Comparison of real and modified lift curves](image)

According to GRAF et al [2] the iterative process needs under-relaxation in order to achieve convergence. Under-relaxation factors of \( \omega = 0.1 \) for the wing and \( \omega = 0.05 \) for the appendages have been selected and used in Eq. (31).

\[
v_{\text{ind}} = \frac{v_{\text{ind}}^{\text{current step}}}{\omega} + \frac{v_{\text{ind}}^{\text{last step}}}{1 - \omega}
\]

(31)

5 HULL RESISTANCE

Hull resistance can be is split into frictional, wavemaking and viscous pressure components. In this study friction is assessed by applying the ITTC’57 friction line using the static wetted surface area.

For the computation of wavemaking drag OLIVER [9] suggested using the theory developed by MICHELL. His approach belongs to the group of slender ship theories to which the hulls concerned in this study seem ideally suited, with slenderness and length to beam ratios of 11.7 and 17.6, respectively. One disadvantage arising from this choice is the neglect of viscous pressure resistance, since this is not captured in the theory. For more details the reader is referred to [10]

MICHELL’s theory is implemented into the potential flow research code *Michlet*, which was developed by CYBERIAD [10].

Figure 5 shows a comparison of predicted resistance using friction line and panel code to experimental towing tank data of a Tornado class catamaran hull. The resistance hump around hull speed is well captured. Although there is generally good agreement between the datasets up to a Froude number of around 0.8, the disparity between calculated wave-making resistance and measured residuary resistance increases with speed. This is in accordance with expectations since viscous pressure resistance is caused by the loss in momentum through friction, whose
share of the overall increases with speed. Hence, viscous pressure resistance will increase with speed as well. This relationship is also indicated by the shape of the curves with the predicted wave-making drag exhibiting asymptotic behaviour at high Froude numbers, while the residuary resistance values seem to follow the frictional resistance graph.

![Graph showing comparison of predicted and experimental resistance data](image)

**Figure 5:** Comparison of predicted and experimental resistance data

Similar to the hydrostatic forces, the resistance of the hull has to be known as a function of the draft in order to be able to model the yacht’s transition to foiling. Hence resistance curves have been calculated for different drafts. The results are shown in figure 6 for a range of drafts between 0.05 m and 0.45 m.

![Graph showing hull resistance curves for range of drafts](image)

**Figure 6:** Hull resistance curves for range of drafts

### 6 RESULTS

The models have been integrated into a VPP which has been implemented as a constraint optimisation routine in Matlab. Figure 7 shows a polar plot of the VPP outputs for 4 different wind speeds. It can be seen that speeds of the order of up to 2.5 times wind speed are reached.
It can be seen that an increase in wind speed and true wind angle creates the desired effect of an increase in boat speed. This trend continues until the wing lift cannot be increased by an increase in angle of attack anymore, or the minimum heeling moment constraint cannot be satisfied anymore. Furthermore, the transition to foiling can be clearly identified as distinct jumps in speed in two of the curves.

![Speed polar plot for windspeeds of 5.0m/s (black), 7.5m/s (green), 10.0m/s (red) and 12.5m/s (blue)](image)

**Figure 7:** Speed polar plot for windspeeds of 5.0m/s (black), 7.5m/s (green), 10.0m/s (red) and 12.5m/s (blue)

### 7 CONCLUSIONS

- Different models aimed at the calculation of forces for the velocity prediction of catamarans have been presented.
- The results of the velocity prediction are encouraging for further research.
- The hardware demands of all models are such that they can be run on standard desktop computers and still deliver results within a fraction of a second.
- Apart from Matlab, only free software is used, limiting the required resources required to achieve useful VPP predictions.
REFERENCES


SIMULATION-DRIVEN DESIGN OF SAILING YACHTS AND MOTOR BOATS

Bodo Hasubek*, Stefan Harries†

* Dreamcatcher One
Sonnenblumenring 39, 86415 Mering, Germany
info@dreamcatcherone.de, www.dreamcatcherone.de

† FRIENDSHIP SYSTEMS AG
Benzstr. 2, 14482 Potsam, Germany
harries@friendship-systems.com, www.friendship-systems.com

Key words: Simulation-Driven Design (SDD), Yacht Design, Computational Fluid Dynamics (CFD), Parametric Modeling, Optimization

Abstract. The design of yachts and boats can significantly benefit from simulation-driven design (SDD) using codes of Computational Fluid Dynamics (CFD). In SDD a large number of virtual prototypes is investigated numerically for key objectives. In hydro- and aerodynamics objectives often relate to resistance and lift which govern the performance of both sailing yachts and fast motor boats. In order to reduce the dimensionality of the design space, i.e., the degrees-of-freedom, a parametric approach is utilized. For the flow simulation different levels of fidelity are used, ranging from potential flow analysis to viscous flow simulation solving the RANS equations. Design examples applying the SDD approach will be presented for both a sailing yacht and a motor boat. The sailing yacht is a 20m catamaran for worldwide travel and the motor boat is a 6m planing boat for day cruises. Parametric models for the two vessels will be discussed, comprising the generation of surfaces and watertight tri-meshes, the latter of which can be fed to the CFD code of choice. Here SHIPFLOW® and FINE™/Marine were applied in connection with CAESES® which provided both the shapes and the integration of CFD for SDD. To close the simulation driven design cycle of the sailing catamaran an appended version of the parametric model with rudders and daggerboard is used for virtual tank testing. Combining these results with a suitable sail model allows for an accurate velocity prediction (VPP) in an early design stage.

1 Introduction

While CFD ship hull improvements and optimizations are quite common for large commercial vessels, CFD driven design for sailing yachts and small motor boats is usually limited to multi-million dollar projects such as the America’s Cup. Costs are assumed to be prohibitive and incentives are missing. Fuel consumption is not an issue for pleasure boats and sailing performance is attributed to a “heavy” design or to the lack of abilities of the skipper. However, recent developments in parametric design of ship hulls and affordable CFD computations have changed the playing field. In this paper the investigation of a new design of a 20m sailing multihull (Fig. 1) is presented and the classic Riva Junior fast planing craft is revisited. Both designs have
Bodo Hasubek and Stefan Harries

In common that they use a hard chined hull. Nevertheless, the methods discussed can be readily applied to round-bilge hulls as well.

For the sailing catamaran hard chines were chosen for easy manufacturing in aluminium based on developable surfaces. It is shown that a systematic variation of the hull parameters leads to a significant reduction of total resistance compared to a manually lofted hull. After defining the optimization goal, a fully parametric model is developed for the canoe body of a single hull using CAESES®. For the design selection a multiple stage approach was taken: 4508 models were investigated using the potential flow code of SHIPFLOW® for wave resistance. Subsequently, the most promising parameter ranges were chosen to analyse 521 models with regard to their total resistance using SHIPFLOW®’s zonal approach which combines a potential flow solution for the wave resistance with a viscous resistance solution for the rear half of the hull an the wake. The best design was selected and verified by comparing the resistance curves for the whole operating range using both FLOWTECH’s SHIPFLOW® and NUMECA’s FINE™/Marine. The parametric model then is refined to include two hulls as well as rudders and daggerboards. This model is used to find a suitable distance between the hulls and to generate virtual tank testing inputs for a velocity prediction program (VPP) under different sailing conditions using FINE™/Marine.

The example of the motor boat at planing speed presents an automatic minimization of resistance via a design-of-experiment (DoE) combined with a deterministic search. CAESES® and FINE™/Marine were coupled as an integrated solution (Fig. 2). The example serves to highlight the procedure and the various elements involved – namely variable geometry, high-fidelity CFD and formal optimization – along with improvements that could be achieved.

2 Simulation-Driven Design of a sailing catamaran

2.1 Design Objectives

For motor driven vessels the design objectives are usually provided by the client in terms of pay load and velocity requirements. A sailing yacht, however, operates at an arbitrarily number of load and velocity conditions which cannot be easily predicted. Therefore, for the load condition the worst case scenario of “fully loaded” was chosen at a displacement of 36t. Due to wave theory the total resistance vs. Froude number curve shows a certain “hump” at around \( F_n = 0.3 \). This hump can be seen in the resistance curve of the reference design. For the catamaran with a waterline length of 20 meters \( F_n = 0.3 \) translates into a boat speed of 8.2 knots. Apparently, the speed...
range between 8 and 10 knots should be within reach of a catamaran of this size even in light winds. Therefore, the reduction of this hump was chosen as the primary optimization goal. At the same time the optimization should not sacrifice the speed ability in normal wind conditions. From analytical estimates considering an upwind sail area of 270 m² the normal upwind ability results in around 12 to 13 knots. This leads to a two speeds optimization approach for \( F_n = 0.3 \) (8.2 knots) and \( F_n = 0.44 \) (12 knots). Consequently, all 4508 models were evaluated for these two speeds to select the designs with the best overall performance.

### 2.2 Parametric Models

The parametric models use three sections, an aft, a main and a bow section. A model with three plates and one with four plates below the waterline was developed to investigate whether a better approximation of an optimum elliptical shape by using more plates would lead to better designs. Both section models use given, but variable waterline and keel line curves as well as a flare angle and a dead rise angle curve over the complete hull length. This allows for a flexible positioning of the main sections. The three plate model further uses the sectional areas as input to generate the section.

![Figure 3: Three plate model: section geometry](image)

![Figure 4: Four plate model: section geometry](image)

For reduction of total resistance the wetted surface area needs to be considered as well. In the current approach the length of the section curve (length from \( p_1-p_2-p_3-p_4 \) in Fig. 3) was minimized using a quadratic equation to find the positions of points \( p_2 \) and \( p_3 \) in dependance of the sectional area, deadrise angle, the side angle and the flare angle. Using a Brent minimization approach within CAESES® the side angle was varied to minimize the length \( p_1-p_2-p_3-p_4 \). The idea follows the assumption that if the curve lengths of the sections are minimal and the hard chines connecting the points among the aft, main and bow sections are fair, then the wetted area of the surfaces created between these hard chines is at least close to its minimum as well. Analysing the created models showed that the wetted surface area among all designs varied only by a maximum of 1.5%.

Many hand-made design variations suffer from varying displacement values [1]. In the current parametric approach the main section area is varied using a second optimization to meet the displacement of 18t per hull with a deviation of less than 0.1‰. By keeping the displacement constant any variations in resistance can be directly attributed to the changes in the geometry.

The four plate model (Fig. 4) follows a similar approach whereby the geometry depends on \( r_1 = T_c \), \( r_2 \), \( r_3 \), and \( r_4 \).
$r_1 = BWL$ as well as dead rise and flare angle. The radius $r_2$ halves the angle between $r_1$ and $r_3$ and is chosen such that the length $p_0-p_1-p_2-p_3-p_4$ is reduced. The surface between $p_3$ and $p_4$ is extended linearly up to a further chine well above the waterline to avoid the introduction of an additional chine at the waterline. A hull generated using the four plate model is shown in Fig. 5. The waterline is marked in red.

The models comprise more than 20 parameters completely defining the hull’s shape. Seven to eight of these parameters (depending on the model three vs. four plates) have been systematically varied while the remaining parameters were fixed to meaningful values determined in a preliminary evaluation phase. Compared to conventional spline-based approaches using an uncontrollable large number of vertices the number of parameters is very low, but still allows for a wide range of possible hull shapes. It should be mentioned that not all parameter combinations lead to valid hull shapes: Out of 6000 parameter combinations for the potential flow analysis only around 4500 resulted in a feasible hull shapes. The number of feasible designs within a parameter range is an indicator for the stability of the geometry of a parametric model. The larger the allowable range for each parameter and the more extreme the allowable parameter combinations are the more flexible the model is. This is particularly important if non-conventional hull shapes are of interest. It was found that the parameters for bow fineness and stern width showed highest sensitivity towards the optimization goals. It also seems that the alignment of the hard chines with the flow around the hull strongly influences the performance. This does not come as a big surprise. However, an alignment of the chines with the flow does not lend itself to easily accessible design parameters without violating the fairness of the lines. Therefore, only with a systematic search the most favourable designs could be found.

2.3 CFD Analysis

The main advantage of a parametric model created in CAESES® is its universal applicability to different CFD codes such as SHIPFLOW® and FINE™/Marine. For SHIPFLOW® a station based geometry export was used while FINE™/Marine requires tessellations of the complete calculation domain in the form of multi-body STL files. These simulations can be highly automated. The simulation times vary greatly from around 30 minutes for a wave resistance calculation using SHIPFLOW®’s potential flow solver, via around 4 hours for SHIPFLOW®’s zonal approach, both on a conventional 3.5GHz 4 core PC with 16Gb RAM, all the way up to more than 30 hours on a 3.0GHz 8 core PC with 64Gb RAM for a viscous resistance calculation with a free surface of an appended two-hull configuration using FINE™/Marine.

For the selected designs their total resistance over the complete operation speed range was calculated to verify that the lower resistance at $F_n = 0.3$ is not penalized at other velocities. Fig. 6 shows that the resistance hump at $F_n = 0.3$ could be mostly eliminated without sacrificing performance at higher speeds. The four plate model even performs better at higher speeds than the reference design.

In table 1 the numerical result as well as some geometrical and hydrostatic parameters of the designs are summarized. The main objective of reducing the total resistance at $F_n = 0.3$ was well achieved. The best three plate design shows a total resistance reduced by 13.5%, the best four plate design shows a reduction of
At the same time the performance at $F_n = 0.44$ did not suffer and even slightly decreased as well for the latter. Moreover, an interesting result is that beyond an L/B ratio of 10 it is not guaranteed that a larger L/B value stands for better performance. Figures 8 and 9 show the wave patterns of the best three and four plate designs compared to the reference design at $F_n = 0.3$ (8.2 knots) and 0° heel. The improvements in the wave patterns show at the fore part of the hull and in the wave elimination in the wake.

### Performance Parameters

<table>
<thead>
<tr>
<th></th>
<th>Reference Design</th>
<th>Three Plate Design</th>
<th>Four Plate Design</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R_t (F_n = 0.3)$ [N]</td>
<td>1723</td>
<td>1490</td>
<td>1428</td>
</tr>
<tr>
<td>$R_t (F_n = 0.44)$ [N]</td>
<td>5277</td>
<td>5274</td>
<td>5195</td>
</tr>
<tr>
<td>Trim at $F_n = 0.44$ [°]</td>
<td>0.82</td>
<td>0.83</td>
<td>0.86</td>
</tr>
</tbody>
</table>

### Hull Parameters

<table>
<thead>
<tr>
<th></th>
<th>Reference Design</th>
<th>Three Plate Design</th>
<th>Four Plate Design</th>
</tr>
</thead>
<tbody>
<tr>
<td>LWL [m]</td>
<td>20.1</td>
<td>20.1</td>
<td>20.17</td>
</tr>
<tr>
<td>BWL [m]</td>
<td>1.60</td>
<td>1.90</td>
<td>1.84</td>
</tr>
<tr>
<td>$T_c$ [m]</td>
<td>1.05</td>
<td>1.02</td>
<td>0.97</td>
</tr>
<tr>
<td>Wetted surface [m²]</td>
<td>46.7</td>
<td>46.1</td>
<td>46.4</td>
</tr>
<tr>
<td>L/B</td>
<td>12.6</td>
<td>10.6</td>
<td>10.95</td>
</tr>
<tr>
<td>BTR</td>
<td>1.52</td>
<td>1.87</td>
<td>1.9</td>
</tr>
<tr>
<td>$C_p$</td>
<td>0.634</td>
<td>0.588</td>
<td>0.595</td>
</tr>
</tbody>
</table>

Table 1: Numerical CFD results and hull parameters of the demi-hull

In order to verify the validity of the results, the resistance curve of the best four plate design was re-evaluated with a different CFD code (NUMECA’s FINE™/Marine) which uses a free-surface RANSE approach (Fig. 7). The maximum deviation between calculated values lies at $F_n = 0.36$ and reaches 5.5%, while at the optimization points $F_n = 0.3$ and $F_n = 0.44$ the deviation is 1.8% and 0.1%, respectively. Therefore, the optimization procedure can be assumed to deliver reliable results within an acceptable margin of error for a cruising catamaran. It is noted that the relative positioning of the different designs with regard to their performance is usually even more reliable than their absolute resistance value if the comparative evaluations are performed using the same mesh resolution and the same solver (as was done here).
Figure 8: Demi-hull at $F_n = 0.3$: Reference design vs. best three plate design at 0° heel

Figure 9: Demi-hull at $F_n = 0.3$: Reference design vs. best four plate design at 0° heel
Bodo Hasubek and Stefan Harries

One hull vs. Two hulls catamaran configuration

<table>
<thead>
<tr>
<th>$F_n$</th>
<th>Resistance [N]</th>
<th>BCB [m]</th>
<th>Resistance</th>
<th>$\Delta R$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.3</td>
<td>2872</td>
<td>4.25</td>
<td>2908</td>
<td>1.3%</td>
</tr>
<tr>
<td>0.44</td>
<td>10568</td>
<td>4.00</td>
<td>12908</td>
<td>22.1%</td>
</tr>
<tr>
<td>0.44</td>
<td>10568</td>
<td>4.25</td>
<td>12506</td>
<td>18.3%</td>
</tr>
<tr>
<td>0.44</td>
<td>10568</td>
<td>4.50</td>
<td>12258</td>
<td>16.0%</td>
</tr>
</tbody>
</table>

Table 2: Resistance of one hull vs. two hulls catamaran configuration

2.4 Closing the design loop

A further advantage of a parametric model is the ability to re-use all the properties of the single hull canoe body model and refine it to include a double hull setup and appendages (rudders and daggerboards). Such a refined model is used to determine the best compromise between the hull’s centerlines distance (BCB) and total resistance.

Table 2 shows the influence of a single hull compared to a two hull catamaran configuration. It can be seen that at $F_n = 0.3$ there is close to no interference between the hulls, while at $F_n = 0.44$ the hull interaction significantly increases the total resistance. The wave interaction for $F_n = 0.44$ is shown in figures 10 and 11. As expected, the data show that increasing BCB reduces the total resistance, but the effect slows down for larger BCBs. Therefore, a BCB of 4.25 meters was taken as a compromise.

For a preliminary performance prediction of the catamaran design the fully appended parametric model is used to generate the resistance data required for a velocity prediction program (VPP). Common VPP packages rely on hull data that is derived from the Delft systematic series with additions for heel and appendages. While this works satisfactorily for monohull designs with rounded sections, the data basis of the Delft series does not support hard chined multihull designs very well. Furthermore, sailing catamarans with daggerboards are normally sailed with the leeward daggerboard down only. This results in a highly asymmetric setup that is not easily accessible to analytical approaches.

This problem is overcome by calculating the resistance in the direction of movement and the sideforces generated by the hull and appendages for heel angles from 0° to 5°, leeway angles between 0° and 4° and for a velocity range between 4 and 14 knots. The limits were chosen in accordance with the anticipated operational profile. For example, the maximum heel was limited 5° since this represents the angle at which half the maximum static stability is reached. A cruising catamaran should not be sailed beyond this point for safety reasons.

With four points in each of these dimensions this approach leads to 64 calculations in total. Fig. 12
Figure 12: Water tight triangulation of the appended model shows the watertight triangulation of the asymmetrically appended parametric hull model. The asymmetry requires a complete calculation domain of around 6 million cells. The calculations were performed on said 8 core PC. They took 77 days. Obviously this time frame needs to be reduced for every day use. However, high performance computer clusters with all the software required pre-installed are available for rent and can perform such calculations within a few days [5].

The resulting forces of the CFD calculations depend on the three independent variables heel angle, leeway angle and velocity. Fig. 13 and 14 show an example of the resistance and side forces for a fixed heel angle of 2.5°. For further processing in a VPP these values are interpolated using a spline based scattered data interpolation [2]. In previous work [3] Hazen’s historic sail model is used to calculate the equilibrium of the hydrodynamic and the aerodynamic forces to derive the sailing performance under different sailing conditions. In the current case a more accurate sail model based on the ORC VPP aerodynamic model [4] is employed. Based on the CFD data and the aerodynamic model the expected sailing performance can be predicted accurately in an early design stage.

3 Simulation-Driven Design of a fast planing motor boat

3.1 Design Objective

A small hard-chine motor boat of around 6 m length and 1.3m³ displacement, typically used as a day cruiser on large lakes and as a tender for superyachts in coastal areas, was studied with regard to its performance at a design speed of 18 kn. With a Froude number of 1.32, based on the length of the design waterline at rest, this represents a planing hull for which dynamic trim and sinkage (or rather lift) and the influence of the free surface are crucial.
For this type of boats model test campaigns are often prohibitively expensive. A standard approach therefore is to rely on series data found in literature, see [6, 7]. Once the main particulars are set, the influence of small changes for the better or worse cannot be derived from the series data. Alternatively, a suitable parent hull is taken, slightly modified and subsequently built – naturally, hoping for reasonably good performance.

Both design approaches cannot provide answers to the typical design questions of how deadrise, rocker, hollowness of the planing surface, its warp towards the transom, width and shape of the spray rail and other design parameters should be chosen. In this situation an extended investigation based on simulation-driven design can help. To this effect, the geometry of the motor boat was parametrically modeled within CAESES® while the flow was computed with the high-fidelity code FINE™/Marine, coupled to CAESES® to be automatically executed in a design loop.

### 3.2 Parametric Model

As it is common in design work, an attractive and relevant baseline was chosen as a good starting point. Here the Riva Junior, a motor boat of classic beauty from the mid 1960s, was selected. A fully-parametric model was developed within CAESES® [8]. For the purpose of the study the parametric model was built to support wide changes to the hull while being able to closely approximate the baseline.

![Figure 13: Resistance forces at heel = 2.5°](image1)

![Figure 14: Side forces at heel = 2.5°](image2)

![Figure 15: Building pattern of the generic section (left: section in the aft body, right: section in the forebody](image3)
Similar to the parametrization of the sailing yacht as shown in figures 3 and 4, a building pattern for a generic section is defined first, representing the shapes found from stem to stern. For the motor boat, this section features a curved portion from the keel line to the inner knuckle of the spray rail, a flat portion forming the spray rail itself and another curved portion from the outer knuckle of the spray rail up to the sheer. Fig. 15 illustrates the building pattern for the aft- and forebody, respectively. Note that the actual geometry of the two highlighted sections (shown in red in Fig. 15) differs while the topology remains the same. Longitudinal curves define the parametric input to the building pattern at each position.

Fig. 16: Selected parameters of the fully parametric model

Fig. 16 depicts important design parameters, namely beam, length (from the transom to the peak) and draft to define the main dimensions plus selected parameters for fine-tuning such as rocker (i.e., the rise of the keel line from maximum draft towards the transom stern), the width of the spray rail at the transom, the deadrise angle and the hollowness of the planing surface (i.e., its deviation from the straight connection from the keel to the spray rail). The parametric model is explained in detail in [5].

3.3 CFD analysis

Since viscous and free-surface effects both play important roles for fast planing hulls, FINE™/Marine was selected as a high-fidelity CFD code. The so-called “C-Wizard” by NUMECA was utilized which supports the user in selecting reliable settings for the simulation at hand. While CAESES® provides the boundaries of the flow domain including the hull itself in discretized form (via tri-meshes in STL-format), FINE™/Marine takes care of the grid generation (via HEXPRESS) and the actual flow analysis. Details of the pre-processing, the grid generation and the flow analysis are reported in [5].

During the simulations all boats were free to adjust to the equilibrium positions of the forces in longitu-
dinal and vertical direction and the moments about the transversal axis. The propulsion system, i.e., shaft, bracket, propeller and rudder, was not explicitly modeled but taken into consideration via a thrust force collinear with the inclined shaft line. The flow solver automatically balances the virtual thrust with the total resistance the vessel encounters, yielding the calm-water resistance at 18 kn.

![Designs investigated during the optimization (excerpt)](image)

### 3.4 Closing the design loop

The advantage of investigating a large set of similar designs over modeling and analyzing just one single design (or one design plus a handful of variants) is to substantially increase the probability of identifying a highly effective design. Furthermore, it allows to study the cause-and-effect of geometry on flow fields.

Rather than undertaking the tedious work of remodeling, ex- and importing files, generating meshes, running the CFD analysis and producing plots and diagrams for many designs manually, a closed design loop is called for that, once established, runs these steps automatically. Similar to what was reported above for the sailing yacht, CAESES® was employed to run a design-of-experiment, comprising 100 variants, and
several additional searches, for instance via a T-search which is a deterministic gradient-free search strategy.

Fig. 17 illustrates selected results taken from the large set of designs. It is an excerpt of five arbitrary designs for which trim (top row), pressure distribution on the planing surface (second row), the wave field in top view (third row), the actual geometry (fourth and fifth row) and the wave field in perspective view (final row) are shown – all of which were generated automatically and in exactly the same way, avoiding errors often found in manual processes.

From the many variants a particular effective design was found, namely a design which features 7% less resistance than the baseline, i.e., a reduction from 1572 N to 1461 N at 18 kn. Taking into consideration that the baseline was carefully designed already, this can be regarded as a significant improvement.

4 Conclusions

It was shown that simulation-driven design using fully parametric models leads to more effective yacht and boat designs. The effort is significantly reduced by automating the simulation procedure. CAESES® multiple interfaces to different CFD programs leaves the choice of the most suitable CFD solver to the user and even allows for effortless switching between CFD packages. Fully automatic identification of a design from a multi-dimensional design space always is a challenge because the parameter-to-objective relationships are often complex. Looking at many variants certainly helps to develop a good appreciation of possible compromises and gains. Whenever performance is critical, like velocity-made-good for a sailing yacht, speed attainable for a motor boat or even cruising range for a battery powered craft, a design largely benefits from more an effective hull.

REFERENCES

[1] Perret, Clementine Design and optimization of a 50’ sailing catamaran, Chalmers University of Technology, Department of Shipping and Marine Technology, Gothenborg


ON THE NUMERICAL CONVERGENCE PROPERTIES OF THE CALCULATION OF THE FLOW AROUND THE KVLCC2 TANKER IN UNSTRUCTURED GRIDS

Ana Luísa Rocha*, Luís Eça* and Guilherme Vaz†

*Instituto Superior Técnico
Universidade de Lisboa
Av. Rovisco Pais, 1049-001 Lisboa, Portugal
e-mail: ana.luisa.rocha@tecnico.ulisboa.pt
e-mail luis.eca@tecnico.ulisboa.pt

† Maritime Research Institute Netherlands (MARIN)
2, Haagsteeg
6708 PM Wageningen, The Netherlands
e-mail: G.Vaz@marin.nl

Key words: RANS, Numerical Error, Unstructured Grids, KVLCC2

Abstract. This paper addresses the estimation of numerical errors in the calculation of the flow around the KVLCC2 tanker at model scale Reynolds number in unstructured grids. The flow solution is based on the Reynolds-Averaged Navier-Stokes equations supplemented by the $k-\omega$ SST two-equation eddy-viscosity model using the so-called double-body approach, i.e. free surface effects are neglected. Grid refinement studies are performed for sets of grids generated with the open source code SnappyHexMesh and with the HEXPRESS™ grid generator. Definition of grid refinement ratio in unstructured grids and its consequences for the estimation of numerical errors is discussed. Friction and pressure resistance coefficients and mean velocity components at the propeller plane are compared with reference solutions obtained in nearly-orthogonal multi-block structured grids with the same flow solver ReFRESCO.

1 INTRODUCTION

Calculation of viscous flows around ships at model and full scale Reynolds numbers based on the Reynolds-Averaged Navier-Stokes equations has become standard practice for many CFD flow solvers. The two main challenges that such calculations present are: the ability to reduce the numerical error (iterative and discretization errors) to negligible levels and the selection of the turbulence model that leads to the smallest modelling errors. Recently, a Verification and Validation study has been performed for the KVLCC2 tanker [1] which is a ship geometry that has been thoroughly studied since the 2000 Gothenburg Workshop [2].

In [1], thirteen different turbulence models including one-equation and two-equation eddy-viscosity models and Explicit Algebraic Reynolds Stress Models are tested neglecting free surface
effects and applying no-slip conditions without wall functions. Discretization errors are estimated with grid refinement studies [4] using sets of nearly-orthogonal multi-block structured grids generated with GridPro [3]. In all the simulations presented in [1], it was possible to reduce the iterative error\(^1\) to negligible levels when compared to the discretization error. However, as the geometric complexity of the ship increases, as for example with the use of an energy saving device in the JAPAN Bulk Carrier proposed for the 2015 Tokyo Workshop [5], there is a natural trend to use unstructured grids [6].

Generation of unstructured grids has significantly progressed in the last two decades with several open-source and commercial codes available. There are (at least) two common features in most of the viscous flow simulations performed in unstructured grids: the use of near-wall layers of cells that intend to preserve grid orthogonality in the near-wall region and the use of local refinement boxes. This latter feature can originate the so-called hanging nodes that may affect the iterative and discretization errors of the numerical solutions. Therefore, it is important to investigate the impact of the use of unstructured grids on the numerical accuracy of ship viscous flows. To this end, we have selected the flow around the KVLCC2 tanker\(^2\) at model scale Reynolds number with the same domain size and boundary conditions used in the study presented in [1]. Our goal is twofold: address iterative and discretization errors of RANS solutions performed in unstructured grids generated with two different grid generators: an open source code SnappyHexMesh [7] and the commercial package HEXPRESS\(^{TM}\) [8]; investigate the use of grid refinement studies and power series expansions to estimate discretization errors for unstructured grids.

Although the results presented in [1] show that discretization errors for a given grid depend on the selected turbulence model and quantity of interest, in the present exercise, we have restricted ourselves to the shear-stress transport (SST) \(k-\omega\) two-equation eddy-viscosity model [9]. Furthermore, the quantities of interest selected for this study are only the friction and pressure resistance coefficients and the mean flow axial velocity component at the locations of the propeller plane that have experimental measurements available [10].

It must be mentioned that generating grids with SnappyHexMesh for the calculation of the flow around the KVLCC2 tanker without wall functions is not a trivial task, even for model scale Reynolds number. Although we can not guarantee that it is impossible to keep the dimensionless near-wall cell height below 1 (\(y_{nw}^+ < 1\)), all the attempts made were unsuccessful. Furthermore, it is not easy to keep \(y_{nw}^+\) above 50 for the complete surface of the ship and so calculations for grids generated with SnappyHexMesh had to be performed with the so-called Automatic Wall Functions, which blend the linear sub-layer with the log-law region [11]. Therefore, we have generated two sets of grids\(^3\) with HEXPRESS\(^{TM}\): one with a near-wall cell size similar to the SnappyHexMesh grids and a second set with (\(y_{nw}^+)_{\text{max}} < 1\). The first set of grids provides a comparison to the SnappyHexMesh results, whereas the second set is compared to the results presented in [1] for the same problem and flow solver ReFRESCO [12].

The paper is organized in the following way: section 2 presents the problem definition, whereas section 3 presents the sets of unstructured grids used in this study and the determination of the

\(^1\)Iterative convergence required maximum normalized residuals of all transport equations solved below \(10^{-8}\). Residuals of transport equations are equivalent to changes of dimensionless variables in a simple Jacobi iteration.

\(^2\)Defined by the same stl file used in [1].

\(^3\)The original plan was to use only open source codes.
grid refinement ratio (typical cell size); the results are presented and discussed in section 4 and the conclusions are summarized in section 5.

2 PROBLEM DEFINITION

2.1 Mathematical Model

In the present work we have adopted the time-averaged continuity and Navier-Stokes (RANS) equations for an incompressible fluid \( \rho = \text{const.} \) supplemented with the SST two-equation \( k - \omega \) eddy-viscosity model described in [9, 13]. It should be mentioned that the production term of the \( k \) transport equation is limited to 15 times dissipation \( \varepsilon = \beta^* k \omega \), which is slightly different from the 10 times \( \varepsilon \) suggested in [9]. However, the purpose of this limiter is to avoid an unphysical overshoot of \( \nu_t \) at the stagnation region and tests performed for a flat plate flow [14] show that the limiter with 10 times \( \varepsilon \) is also active in several regions of the boundary-layer.

2.2 Calculation Domain

The domain for the calculation of the flow around the KVLCC2 tanker at model scale Reynolds number is exactly equal to that adopted in [1]. It is a prismatic rectangular region defined in a Cartesian coordinate system \((x, y, z)\) with the \( x \) axis coincident with the keel line (pointing to the bow), the transverse \( y \) axis perpendicular to the symmetry plane of the ship and the vertical \( z \) axis forming a right-handed system. The calculation domain is illustrated in figure 1.

\[ \text{Figure 1: Illustration of the calculation domain of the flow around the KVLCC2 tanker.} \]

The Reynolds number \( Re \) based on the incoming flow velocity \( V_\infty \) and the distance between perpendiculars \( L_{PP} \) is equal to

\[ Re = \frac{V_\infty L_{PP}}{\nu} = 4.6 \times 10^6. \]

It must be mentioned that the KVLCC2 surface definition is based on the same stl file used in [1], which includes a fairing of the transom of the stern. This small change of the original

\[ ^4 \text{The origin of the coordinate system is on the symmetry plane of the ship at the intersection of the keel line and aft perpendicular.} \]
geometry avoids iterative convergence problems due to detached flow separation regions and the occurrence of vortex shedding.

2.3 Boundary Conditions

The computational domain is bounded by the following surfaces: the surface of the ship and its symmetry plane \((y = 0)\); the still water\(^5\) plane located at \(z = 0.065L_{PP}\); the inlet and outlet planes located at \(x = \pm 2L_{PP}\); the lateral plane located at \(y = L_{PP}\) and the bottom plane placed at \(z = -L_{PP}\).

Velocity components are set equal to undisturbed flow conditions at the inlet, \(x = 2L_{PP}\), i.e. \(V_x = V_\infty, V_y = 0\) and \(V_z = 0\) and the pressure is extrapolated from the interior of the domain. Turbulence kinetic energy is specified from a turbulence intensity of \(I = 1%\) and the \(\omega\) value is specified to obtain \(\nu_t = 0.1\nu\). Streamwise derivatives of all dependent variables are assumed to be zero at the outlet \(x = -2L_{PP}\). Symmetry conditions are applied at the still water plane \((z = 0.065L_{PP})\) and at the symmetry plane of the ship, whereas free slip conditions are applied at the bottom boundary \(z = -L_{PP}\). Pressure is fixed at the lateral boundary \((y = L_{PP})\) and transverse velocity derivatives of the remaining dependent variables are assumed to be zero.

At the ship surface, velocity components are set equal to zero due to impermeability and the no-slip condition and the pressure derivative in the normal direction to the surface is set equal to zero. \(k\) is set equal to zero at the ship surface and \(\omega\) is specified at the near-wall cell centre using the blend of the linear sub-layer and log-law regions proposed in [11]. The determination of the shear-stress at the wall \(\tau_w\) depends on the height of the near-wall cells, i.e. on \(y^+_nw\). For the HEXPRESS\(^TM\) grids (see section 3.2) with \(y^+_nw < 1\), \(\tau_w\) is determined directly from its definition, whereas for all the other grids used in this study the velocity profile proposed in [11] is used to determine \(\tau_w\) and the production of \(k\) in the near-wall cell is based on \(\tau_w\).

3 GRID GENERATION

The original goal of this study was to generate unstructured grids using only the open source grid generator SnappyHexMesh [7]. However, a mandatory condition was to be able to generate grids suitable for the application of the no-slip condition without the use of wall functions. Unfortunately, we have not succeeded in fulfilling this requirement with SnappyHexMesh and so we have also used the commercial grid generator HEXPRESS\(^TM\) to generate sets of unstructured grids that exhibit \(y^+_nw < 1\). On the other hand, results obtained in the SnappyHexMesh grid sets cannot be directly compared with the results given in [1] due to the change in the calculation of the shear-stress at the wall (with or without wall functions). Therefore, we have generated a second grid set with HEXPRESS\(^TM\) that presents similar values of \(y^+_nw\) to those obtained in the SnappyHexMesh grids.

3.1 SnappyHexMesh Grids

SnappyHexMesh is an unstructured grid generator that is included in the open source CFD software OpenFOAM [7]. The grid generation process in the SnappyHexMesh utility is done in three consecutive steps: creation of an hexahedral parametric grid covering the whole domain

\(^5\)Free surface effects are neglected.
without the ship surface using the BlockMesh utility; introduction of the ship geometry defined with a triangulated surface geometry file in Stereolithography (STL) format; insertion of near-wall cell layers that provide the required near-wall cell height for the application of the no-slip condition.

The most delicate step of the procedure is the latter. There are several utilities for layer refinement within OpenFOAM [7]. However, we have used SnappyHexMesh for both layer addition and refinement, because all other alternatives lead to unacceptable cells (negative Jacobian), especially at the stern of the ship.

The grid generation procedure adopted tried to keep geometrical similarity of the grids as much as possible. To this end, the twelve different blocks defined in BlockMesh (step 1) were systematically refined in the $x$, $y$ and $z$ directions to create a set of eight basis grids. In the second step of the procedure that iteratively conforms the grid to the 3-D geometry of the KVLCC2 hull, we have adopted two different approaches: for grid set S1 a single refinement box with an extra refinement level of three is set around the whole hull; extra refinement boxes were added at the bow and stern of the ship for grid set S2. As mentioned above, none of the strategies tested enabled the generation of grids with $y^+_{nw} < 1$ using the tools available in the OpenFoam [7] toolbox. Therefore, we generated grids appropriate for the application of the no-slip condition with wall functions, i.e. $y^+_{nw} > 30 – 50$. This was accomplished for the eight grids of sets S1 and S2 using a trial and error approach. As a consequence, the two grid sets present two main difficulties: the insertion of near-wall cell layers must be tuned grid by grid and so geometrically similarity is naturally destroyed; it is not possible to keep $y^+_{nw} > 30 – 50$ for the complete ship surface and so “automatic” wall functions [11] must be used for the application of the no-slip condition.

Table 1: Number of interior cells $N_c$, number of cells on the ship surface $N_s$ and average ($y^+_{nw})_{avg}$, maximum ($y^+_{nw})_{max}$ and minimum ($y^+_{nw})_{min}$ values of the dimensionless near-wall cell height of the eight grids of sets S1 and S2 generated with SnappyHexMesh.

<table>
<thead>
<tr>
<th></th>
<th>$N_c \times 10^{-6}$</th>
<th>$N_s \times 10^{-5}$</th>
<th>$y^+_{nw}$ avg</th>
<th>$y^+_{nw}$ max</th>
<th>$y^+_{nw}$ min</th>
</tr>
</thead>
<tbody>
<tr>
<td>S1_1</td>
<td>22.2</td>
<td>5.52</td>
<td>45.3</td>
<td>72.5</td>
<td>0.98</td>
</tr>
<tr>
<td>S1_2</td>
<td>15.8</td>
<td>4.36</td>
<td>45.2</td>
<td>72.2</td>
<td>1.00</td>
</tr>
<tr>
<td>S1_3</td>
<td>11.1</td>
<td>3.34</td>
<td>46.0</td>
<td>110.0</td>
<td>1.65</td>
</tr>
<tr>
<td>S1_4</td>
<td>7.12</td>
<td>2.46</td>
<td>46.0</td>
<td>72.6</td>
<td>1.58</td>
</tr>
<tr>
<td>S1_5</td>
<td>4.48</td>
<td>1.71</td>
<td>46.0</td>
<td>129.0</td>
<td>2.19</td>
</tr>
<tr>
<td>S1_6</td>
<td>2.62</td>
<td>1.09</td>
<td>47.4</td>
<td>188.0</td>
<td>1.20</td>
</tr>
<tr>
<td>S1_7</td>
<td>1.30</td>
<td>0.62</td>
<td>46.3</td>
<td>254.0</td>
<td>3.41</td>
</tr>
<tr>
<td>S1_8</td>
<td>0.86</td>
<td>0.42</td>
<td>49.3</td>
<td>428.0</td>
<td>1.04</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>$N_c \times 10^{-6}$</th>
<th>$N_s \times 10^{-5}$</th>
<th>$y^+_{nw}$ avg</th>
<th>$y^+_{nw}$ max</th>
<th>$y^+_{nw}$ min</th>
</tr>
</thead>
<tbody>
<tr>
<td>S2_1</td>
<td>21.8</td>
<td>3.35</td>
<td>40.1</td>
<td>69.1</td>
<td>0.29</td>
</tr>
<tr>
<td>S2_2</td>
<td>15.6</td>
<td>2.65</td>
<td>36.4</td>
<td>78.6</td>
<td>0.18</td>
</tr>
<tr>
<td>S2_3</td>
<td>10.8</td>
<td>2.03</td>
<td>37.1</td>
<td>88.8</td>
<td>0.11</td>
</tr>
<tr>
<td>S2_4</td>
<td>6.91</td>
<td>1.49</td>
<td>35.7</td>
<td>93.2</td>
<td>0.34</td>
</tr>
<tr>
<td>S2_5</td>
<td>4.11</td>
<td>1.03</td>
<td>40.7</td>
<td>81.8</td>
<td>0.38</td>
</tr>
<tr>
<td>S2_6</td>
<td>2.21</td>
<td>0.67</td>
<td>37.0</td>
<td>112.0</td>
<td>0.40</td>
</tr>
<tr>
<td>S2_7</td>
<td>1.05</td>
<td>0.37</td>
<td>42.1</td>
<td>211.0</td>
<td>0.69</td>
</tr>
<tr>
<td>S2_8</td>
<td>0.63</td>
<td>0.26</td>
<td>41.8</td>
<td>190.0</td>
<td>0.58</td>
</tr>
</tbody>
</table>

Table 1 presents the number of interior cells $N_c$, the number of cells on the ship surface $N_s$ and the average ($y^+_{nw})_{avg}$, maximum ($y^+_{nw})_{max}$ and minimum ($y^+_{nw})_{min}$ values of the dimensionless near-wall cell height of the eight grids of sets S1 and S2. Figure 2 presents an illustration of
Figure 2: Illustration of the grids generated with SnappyHexMesh at the bow and stern of the KVLCC2 tanker.

 grids S1 and S2 at the bow and stern regions, where it is visible the poor definition of the ship’s geometry at the stern. On the other hand, the lack of control over the introduction of the near-wall cell layers is exemplified by the oscillations obtained for \((y^+_nw)_{avg}\), \((y^+_nw)_{max}\) and \((y^+_nw)_{min}\). Naturally, the properties of the grids included in these two sets are far from ideal. Nonetheless, they will serve two goals on the present study: investigate the robustness of the flow solver and the estimation of discretization of errors based on power series expansions.

3.2 HEXPRESS™ Grids

The goal of the HEXPRESS™ grid sets in this work is to generate solutions to compare with the results obtained in the SnappyHexMesh sets presented in the previous section and to the results presented in [1] obtained in multi-block structured grids.

HEXPRESS™ generates non-conformal body-fitted hexahedral unstructured grids on arbitrary geometries following a sequence of eight main steps. The present grids were generated with the procedure described in [15] with the hull geometry divided into seven different parts (the bulbous bow, the fore-ship, the bilge, the upper-mid-ship, the lower-mid-ship, the aft-ship and the stern) in order to facilitate the surface refinements and the posterior addition of the prismatic layers.
Figure 3: Illustration of the grids generated with HEXPRESS™ at the bow and stern of the KVLCC2 tanker.

Two sets of six grids were generated with HEXPRESS™: set H1 includes grids with \((y_{nw}^+)^{\text{avg}} \approx 30 - 50\) and set H2 grids with \((y_{nw}^+)^{\text{max}} < 1\). Compared to the grids used in [15], the present grids include extra grid refinement in the near-wake region. Table 2 presents \(N_c\), \(N_s\), \((y_{nw}^+)^{\text{avg}}\), \((y_{nw}^+)^{\text{max}}\) and \((y_{nw}^+)^{\text{min}}\) of the six grids of sets H1 and H2 and figure 3 presents an illustration of grids H1_5 and H2_5 at the bow and stern regions. As expected, the control over the insertion of the near-wall cell layers is significantly better than in SnappyHexMesh.

3.3 Definition of Grid Refinement Ratio \(r_i\)

The use of power series expansions to estimate the discretization error of numerical solutions requires the definition of the typical cell size \(h_i\) to define the grid refinement ratio \(r_i = h_i/h_1\), where \(h_1\) stands for the typical cell size of the finest grid [4]. In sets of unstructured grids, it is not possible to keep strict geometrical similarity and so it is important to check the influence of
Table 2: Number of interior cells $N_c$, number of cells on the ship surface $N_s$ and average ($y_{nw}^+$)avg, maximum ($y_{nw}^+$)max and minimum ($y_{nw}^+$)min values of the dimensionless near-wall cell height of the six grids of sets H1 and H2 generated with HEXPRESS™.

<table>
<thead>
<tr>
<th></th>
<th>$N_c$ x10^{-6}</th>
<th>$N_s$ x10^{-5}</th>
<th>$y_{nw}^+$ avg</th>
<th>$y_{nw}^+$ max</th>
<th>$y_{nw}^+$ min</th>
</tr>
</thead>
<tbody>
<tr>
<td>H1</td>
<td>23.3</td>
<td>9.71</td>
<td>32.4</td>
<td>56.8</td>
<td>0.02</td>
</tr>
<tr>
<td>H1</td>
<td>13.7</td>
<td>6.27</td>
<td>33.6</td>
<td>62.7</td>
<td>0.04</td>
</tr>
<tr>
<td>H13</td>
<td>7.09</td>
<td>3.58</td>
<td>39.7</td>
<td>66.1</td>
<td>0.13</td>
</tr>
<tr>
<td>H14</td>
<td>2.94</td>
<td>1.64</td>
<td>42.6</td>
<td>70.5</td>
<td>0.18</td>
</tr>
<tr>
<td>H15</td>
<td>1.49</td>
<td>0.94</td>
<td>43.9</td>
<td>71.8</td>
<td>0.37</td>
</tr>
<tr>
<td>H16</td>
<td>0.61</td>
<td>0.43</td>
<td>45.3</td>
<td>73.1</td>
<td>0.22</td>
</tr>
</tbody>
</table>

The $r_i$ definition on the estimated uncertainties. In this study, we have determined $r_i$ based on the volume of the grid cells $V$ and on the area of the ship surface cells $S$ using $r_i = (V_i/V_1)^{1/3}$ or $r_i = (S_i/S_1)^{1/2}$.

For a grid with $N_c$ interior cells and $N_s$ cells on the ship surface, four different values of $V$ and $S$ are determined:

1. The mean value of the cells volume $V^1$ and area $S^1$,

$$V^1 = \frac{\sum_{j=1}^{N_c} V_j}{N_c}$$

$$S^1 = \frac{\sum_{j=1}^{N_s} S_j}{N_s}.$$

2. The root mean squared value of the cells volume $V^2$ and area $S^2$,

$$V^2 = \sqrt{\frac{\sum_{j=1}^{N_c} V_j^2}{N_c}}$$

$$S^2 = \sqrt{\frac{\sum_{j=1}^{N_s} S_j^2}{N_s}}.$$

3. The mode of the cells volume $V^m$ and area $S^m$, which corresponds to the values of $V$ and $S$ that occur more often. $V^m$ and $S^m$ are determined from an histogram containing forty intervals of constant width in logarithm scales. The values of $V^m$ and $S^m$ are equal to the mid-point of the interval containing the largest number of cells.

4. The maximum value of the cells volume $V^M$ and area $S^M$, $V^M = \text{MAX}(V_j)$ and $S^M = \text{MAX}(S_j)$.

If the grids of a given set were strictly geometrically similar, the values of $r_i$ based on $V^1$, $V^2$, $V^m$, $V^M$, $S^1$, $S^2$, $S^m$ and $S^M$ would be all equal. In the present grid sets, the values of $r_i$ obtained from the eight alternatives described above are not identical, as illustrated in figure 4.
Although there are significant differences between the results obtained for the SnappyHexMesh and HEXPRESS™ grid sets, there are two common trends in the four grid sets: for a given set, the grids on the ship surface are nearly geometrically similar due to the resemblance of all values based on $S$; the differences between the different values of $r_i$ increase with the grid coarsening.

For the S1 and S2 sets, the two alternatives based on $V$ that lead to the results most similar to those obtained with $S$ are $V^m$ and $V^M$. However, these two alternatives are completely unreliable for the H1 and H2 sets due to the changes in grid topology obtained for the coarsest grids of these sets in the outer parts of the domain. As illustrated in figure 5 for the four coarsest grids of set H2, the largest cells reduce size from $H2_4$ to $H2_5$ and remain almost identical between $H2_5$ and $H2_6$. A similar phenomena occurs for the coarsest grids of the H1 set. Therefore, defining $r_i$ based on $S$ seems to be a better choice than using $V$. Nonetheless, we will compare the estimation of discretization errors using these eight definitions of $r_i$. Naturally, we drop the values of $r_i$ obtained from $V^m$ and $V^M$ for grid sets H1 and H2.

4 RESULTS

4.1 Numerical Details

The calculations performed for the S1, S2, H1 and H2 grid sets were performed with a segregated approach and the diffusion terms of all transport equations approximated by second-
order central differences including non-orthogonality corrections [16, 17]. For the S1 and S2 grids, all calculations were performed with second-order upwind (QUICK) with limiters (QL) in the convective terms of the momentum equations and first-order upwind applied to the convective terms of the $k$ and $\omega$ transport equations. For the H1 and H2 grids two alternatives were tested: second-order upwind without limiters (Q) in the convective terms of the momentum equations; second-order upwind with limiters (QTL) in the convective terms of all transport equations.

The selected quantities of interest are the pressure $C_P$ and friction $C_F$ resistance coefficients and the Cartesian mean axial velocity components $V_x$ at the 674 locations of the propeller plane $x = 0.0175L_{PP}$ with available experimental measurements [10].

4.2 Iterative Errors

The iterative convergence criteria targeted for all the present simulations was a maximum normalised residual ($L_\infty$ norm) for all transport equations of $10^{-8}$. Normalised residuals are equivalent to dimensionless variable changes in a simple Jacobi iteration with $\rho$, $V_\infty$ and $L_{PP}$ used as reference quantities. Not all the simulations were able to attain such level of iterative convergence with residuals stagnating at a higher level than desired in some cases.

Overall, iterative converge is worse for the SnappyHexMesh (S1 and S2) grids than for the HEXPRESS\textsuperscript{TM} (H1 and H2) grids. Only three grids of set S1 converged to the required tolerance (S1\textsubscript{2}, S1\textsubscript{3} and S1\textsubscript{4}) and convergence stagnated for residuals of momentum and continuity equations of the order $10^{-4}$ for grid S1\textsubscript{6} and $10^{-5}$ for the remaining grids of this set. On the other hand, the six coarsest grids of the S2 set satisfy the selected convergence criteria and the calculations in the two finest grids were stopped after $10^4$ iterations with the largest residuals of the order of $10^{-6}$.

Almost all calculations performed with the QL and Q settings in the H1 and H2 grids converge to the selected criteria. Typical convergence histories are illustrated in figure 6 for the coarsest (QL settings) and finest (QL, Q and QTL settings) grids of set H2. As expected, the number of iterations required to satisfy the convergence criteria increases with grid refinement and is smaller for the grids of set H1 than for those of set H2. On the other hand, very similar convergence behaviours are obtained for the QL and Q settings with the exception of the H1\textsubscript{2} grid where QL leads to stagnation of the largest residuals of the momentum equations at $10^{-4}$. However, the root mean squared value of the residuals ($L_2$ norm) dropped to $10^{-7}$. Only one of the calculations performed with QTL satisfied the selected convergence criteria (grid H1\textsubscript{6}).
Figure 6: \(L_\infty\) and \(L_2\) norms of the residuals of the simulations performed in the coarsest (H1\(_6\) and H2\(_6\)) and finest (H1\(_1\) and H2\(_1\)) grids of sets H1 and H2. Second-order upwind with (QL) and without (Q) limiters in the convective terms of the momentum equations and first-order upwind in the convective terms of the \(k\) and \(\omega\) transport equations and second-order upwind with limiters in all transport equations (QTL).

However, as illustrated in figure 6, the residuals of the \(k\) transport equation stagnate (\(L_\infty\) norm of the order of \(10^{-6}\)) but the remaining residuals keep dropping.

### 4.3 Discretization Errors

One of the main goals of the present exercise is to investigate the use of unstructured grids in the estimation of discretization errors with power series expansions [4]. As discussed in section 3.3, we have defined the grid refinement ratio \(r_i\) with eight different alternatives and estimated the uncertainty of the selected quantities of interest using all the values of \(r_i\). As an example of the results obtained, figure 7 presents the friction \(C_F\) and pressure \(C_P\) resistance coefficients as a function of \(r_i\) for grid sets S1, S2, H1 and H2. Each plot presents the lines fitted to the five
Three discretization settings tested: second-order upwind with (QL) and without (Q) limiters in the convective terms of the momentum equations and first-order upwind in the convective terms of the \( k \) and \( \omega \) transport equations; second-order upwind with limiters (QTL) in the convective terms of all transport equations.

The results obtained in the SnappyHexMesh grids exhibit a lot more scatter than the data obtained in the HEXPRESS\textsuperscript{TM} sets. Nonetheless, the fits performed with the several definitions of \( r_i \) tested are all very similar with just one exception for \( C_P \) in the S1 and S2 sets. This is an encouraging result for the estimation of discretization errors in sets of unstructured grids, especially for the H1 and H2 sets that exhibit minor differences in the estimated exact solutions obtained with the six definitions of \( r_i \) tested. In the remaining of the paper, we will use the values of \( r_i \) determined from the average value of the area of the cells on the ship surface \( S^1 \).

Figure 8 presents the convergence of \( C_F \) and \( C_P \) with grid refinement for the H1 and H2 grid sets using the QL, Q and QTL settings. The results show a larger influence of the discretization settings on the force coefficients with the exception of \( C_F \) on the H2 set (no wall functions). The use of second-order upwind in the turbulence model exhibits an influence on \( C_P \) that was not observed in the multi-block structured grids used in [1]. This result suggests that this finest grids of each set with the several definitions of \( r_i \) tested.

Figure 9: Isolines of mean axial velocity \( V_x \) for different levels of grid refinement. Two discretization settings: second-order upwind with limiters (QL) in the convective terms of the momentum equations and first-order upwind in the convective terms of the \( k \) and \( \omega \) transport equations; second-order upwind with limiters (QTL) in the convective terms of all transport equations.
Figure 10: Maximum, average and standard deviation of the numerical uncertainty of the mean axial velocity $V_x$ ($U(V_x)$) at the propeller plane for the finest grid of all sets. Three discretization settings: second-order upwind with (QL) and without (Q) limiters in the convective terms of the momentum equations and first-order upwind in the convective terms of the $k$ and $\omega$ transport equations; second-order upwind with limiters (QTL) in the convective terms of all transport equations.

choice must also have a strong impact on the boundary-layer development, as illustrated by the axial velocity isolines at the propeller plane depicted in figure 9, which were obtained with the QL (left) and QTL (right) settings for three levels of grid refinement. There is a significantly larger influence of the grid size on the QL solutions than on the QTL fields, which exhibit small differences between the $H_1$ and $H_3$ grids of the two sets. Furthermore, there is a significant difference between the isolines obtained in the finest grids of the two sets with the QL and QTL settings.

Figure 10 presents the estimated numerical uncertainties of $V_x$ at the propeller plane for all the finest grids and discretization settings presented in this paper. The plots include the maximum $U(V_x)_{\text{max}}$, average $U(V_x)_{\text{avg}}$ and standard deviation $U(V_x)_{\text{std}}$ at the 640 locations with available experimental data and the percentage of locations that exhibit monotonic convergence $F(V_x)_{\text{mon}}$.

The main trends observed in the data are: the numerical uncertainty for the SnappyHexMesh grids is significantly larger than for the HEXPRESS$^\text{TM}$ grids; uncertainties (and solutions) obtained with the QL and Q options are similar; the use of second-order upwind with limiters in the $k$ and $\omega$ transport equations (QTL) reduces the numerical uncertainty of $V_x$, especially for the calculations performed without wall functions (H2). Therefore, we will compare both flow fields with the solutions reported in [1] in the next section.
4.4 Comparison of Solutions

Figure 11 presents the convergence of the friction $C_F$ and pressure $C_P$ resistance coefficients as a function of $r_i$. Wall functions for SnappyHexMesh (S2) and HEXPRESS™ (H1) and no wall functions for HEXPRESS™ (H2) and the results of [1].

The isolines of mean axial velocity component $V_x$ are compared in figure 12 for eight different cases: experimental data [10]; results reported in [1]; finest grids of the S1, S2, H1 and H2 sets with QL; finest grids of the H1 and H2 sets using QTL; There is (again) an excellent agreement between the H2(QTL) solution and the results reported in [1], which in this case are also similar to the H1(QTL) results. All the solutions obtained with first-order upwind in the convective terms of the $k$ and $\omega$ transport equations (QL) exhibit $V_x$ isolines that are closer to the experimental result than those obtained with the QTL settings. The best comparison with the experimental data is actually obtained for the S2 grid that leads to a very (numerically) inaccurate prediction of the friction resistance coefficient. These results show that numerical errors may lead to a fortuitous improvement of the graphical comparison between experiments and simulations. However, if we take into account the grid sensitivity and numerical uncertainty illustrated in figures 9 and 10, it is clear that the numerical uncertainty of the results obtained with QL is not negligible. Therefore, it is important to further refine the grids of sets H1 and H2 to confirm that the results obtained with the QL settings for a sufficiently refined grid will match those obtained with second-order upwind used in the convective terms of all transport equations.
Figure 12: Isolines of mean axial velocity $V_x$ for different levels of grid refinement. Results corresponding to experimental data [10], multi-block structured grids [1], finest grids of SnappyHexMesh sets S1 and S2 (QL) and HEXPRESS™ H1 and H2 (QL and QTL).

5 CONCLUSIONS

This paper addresses the estimation of discretization errors with grid refinement studies in unstructured grids. The selected test case is the flow around the KVLCC2 tanker at model scale Reynolds number, which is calculated with the Reynolds-Averaged Navier-Stokes equations supplemented with the two-equation SST $k-\omega$ eddy-viscosity model. The computational domain and selected boundary conditions are identical to those used in a similar study performed in multi-block structured grids [1].

Sets of unstructured grids were generated with the open source SnappyHexMesh program and with the commercial package HEXPRESS™. The generation of grids that enable the calculation of the flow around the KVLCC2 tanker without wall functions proved to be too difficult for SnappyHexMesh and so the two grid sets generated for this study require the use of the so-called “automatic” wall functions at the surface of the ship. With HEXPRESS™, we have generated a grid set with near-wall cells similar to those obtained with SnappyHexMesh and a second grid set that enables the calculation of the shear-stress at the wall directly from its definition, i.e. no wall functions.

Four different metrics based on the cells volume and surface area of the cells on the surface of the ship were tested to define the grid refinement ratio $r_i$. The results obtained for the four grid sets tested in this study suggest that the surface area of the cells on the surface of the ship is a better choice to define $r_i$ than the cells’ volume. The error estimates performed with the different ways to define $r_i$ showed that it is possible to make reliable error estimates with power series expansions based on grid refinement studies for unstructured grids.

Calculations performed for the four grid sets showed that iterative convergence is more troublesome for the SnappyHexMesh than for the HEXPRESS™ grids. Therefore, the Snappy-HexMesh grids calculations were all performed with first-order upwind in the convective terms.
of the $k$ and $\omega$ transport equations and second-order upwind with limiters in the convective terms of the momentum equations. On the other hand, for the HEXPRESS$^\text{TM}$ grids, it is possible to converge iteratively flow fields without limiters or using second-order upwind with limiters in the turbulence quantities ($k$ and $\omega$) transport equations.

The results obtained for the selected quantities of interest, $C_F$ and $C_P$ resistance coefficients and $V_x$ at the propeller plane, suggest the following conclusions:

- Results obtained in the SnappyHexMesh grids present a lot of scatter in the convergence with grid refinement. Comparison with HEXPRESS$^\text{TM}$ grids suggests that numerically accurate results require a much larger number of grid cells than the roughly $20 \times 10^6$ cells used in the finest grids of this study.

- Results obtained with first and second-order upwind in the $k$ and $\omega$ transport equations show significant differences, with the exception of $C_F$ for the simulations without wall functions. This trend was not observed in the grid refinement studies performed in multi-block structured grids. Furthermore, numerical uncertainty is significantly larger for the wake fields predicted with first-order upwind than for those obtained with second-order upwind in the $k$ and $\omega$ transport equations. On the other hand, the results obtained with second-order upwind in all transport equations are in excellent agreement with data obtained for the same problem in multi-block structured grids.

- Comparison of the predicted wake fields with the experimental data is affected by error cancelling, i.e. differences between experiments and simulations diminish with the increase of the numerical error.

The results presented in this study suggest that the calculation of ship viscous flows in unstructured grids deserves to be further investigated, especially the effect of the accuracy of the discretization of the convective terms of the transport equations of the turbulence model.

Acknowledgment

The authors would like to thank P. Crepier for his help in setting up the domain and grids generated with HEXPRESS$^\text{TM}$.

REFERENCES


RANS-based full-scale power predictions for a general cargo vessel, and comparison with sea-trial results

A.R. STARKE*, K. DRAKOPOULOS†, S.L. TOXOPEUS* AND S.R. TURNOCK†

* Maritime Research Institute Netherlands (MARIN)
2, Haagsteeg, P.O. Box 28, 6700 AA Wageningen, Netherlands
e-mail: b.starke@marin.nl, www.marin.nl

† University of Southampton
University Road, SO17 1BJ, Southampton, UK
e-mail: drakopuloskostas@gmail.com, www.soton.ac.uk

Key words: CFD, power prediction, full-scale validation

Abstract. Blind self-propulsion predictions for the 2016 LR Workshop on Ship Scale Hydrodynamic Computer Simulation have been carried out to simulate the full scale performance of a self-propelled ship in ballast. The single screw ship of 11542 tonnes had been scanned in drydock so the computational model used the actual as operated hull form. It will be shown that using a hybrid RANS-BEM method, the predicted ship speeds at self-propulsion are over-estimated by 0.17-0.28 knots compared to the trial data. The various aspects that influence the accuracy of the direct prediction of power and RPM using CFD are critically discussed.

1 INTRODUCTION

CFD tools are being used more and more extensively in practical ship hull form design and in the prediction of the power and RPM. An evaluation of the progress in this field can e.g. be found in the assessment of the Gothenburg 2010 Workshop and the proceedings of the subsequent workshop in Tokyo, 2015. The cases in that workshop series, however, are until now restricted to model scale. This is motivated by the lack of publicly available full-scale trial data, including the ship and propeller geometries. In the summer of 2016 Lloyd’s Register announced the organization of a hydrodynamics workshop with blind numerical predictions of ship-scale power. The main objective of the workshop was to compare these results with available sea trials measurements, and to assess and develop the capabilities of the numerical tools at ship scale. In the present paper we will discuss our contributions to that workshop.

2 DESCRIPTION OF THE TEST CASE

The ship under consideration is a general cargo vessel built in 1994 in Poland, with a gross weight of 11542 tonnes. The ship is equipped with one four-bladed, right handed, fixed-pitch propeller. Some of the particulars of the ship and the propeller have been listed in Table 1. The reference location for the propeller centre was 2689 mm ahead of the aft perpendicular and 3147 mm above the base. Immediately preceding the trials the vessel was dry-docked and the hull and propeller cleaned. Following that the hull, rudder and propeller were 3D laser
scanned to obtain an accurate geometric representation of the in-service geometry. However, some modifications to the scanned geometry had to be carried out to make them suitable for viscous-flow computations. Many more details of the test case can be found in [1].

Table 1: Particulars of the ship and the propeller.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>L_{pp}</td>
<td>Length between perpendiculars</td>
<td>138</td>
<td>m</td>
</tr>
<tr>
<td>B</td>
<td>Breadth</td>
<td>23</td>
<td>m</td>
</tr>
<tr>
<td>T</td>
<td>Design draught</td>
<td>8.5</td>
<td>m</td>
</tr>
<tr>
<td>T_f</td>
<td>Forward draught</td>
<td>4.899</td>
<td>m</td>
</tr>
<tr>
<td>T_a</td>
<td>Aft draught</td>
<td>5.597</td>
<td>m</td>
</tr>
<tr>
<td>S</td>
<td>Wetted surface area</td>
<td>3727</td>
<td>m$^2$</td>
</tr>
<tr>
<td>V_s</td>
<td>Design speed</td>
<td>14</td>
<td>kts</td>
</tr>
<tr>
<td>ρ</td>
<td>Water density (at 26.5 °C)</td>
<td>1010</td>
<td>kg/m$^3$</td>
</tr>
<tr>
<td>ν</td>
<td>Dynamic viscosity</td>
<td>0.88397x10^{-6}</td>
<td>m$^2$/s</td>
</tr>
<tr>
<td>D</td>
<td>Propeller diameter</td>
<td>5.2</td>
<td>m</td>
</tr>
<tr>
<td>P(0.7 R)/D</td>
<td>Pitch ratio</td>
<td>0.6781</td>
<td>-</td>
</tr>
</tbody>
</table>

3 COMPUTATIONAL METHODS

The viscous-flow method used for the majority of computation reported in this paper is PARNASSOS, a code developed and used by MARIN and IST [2,3]. It solves the discretised Reynolds-averaged Navier-Stokes (RANS) equations for a steady, 3D incompressible flow around a ship’s hull. Various eddy-viscosity turbulence models are available. For all computations in this paper, the one-equation turbulence model by Menter [4] was used, extended with a correction for the longitudinal vorticity by Dacles-Mariani et al. [5].

The discretisation is of finite-difference type. All terms in the momentum and continuity equations are discretised by second or third-order accurate difference schemes. PARNASSOS can handle body-fitted, generally non-orthogonal HO-type grids, either single or multi-block structured.

The momentum and continuity equations are solved in fully coupled form. Therefore, the continuity equation need not be recast in a pressure correction or pressure Poisson equation, but can simply be solved as it is. After discretisation and linearisation, the three momentum equations and the continuity equation give rise to a matrix equation containing 4*4 blocks, which is solved using preconditioned GMRES. This fully coupled solution has been found to be robust and quite insensitive to the mesh aspect ratio. This allows solving the discretized equations on extremely contracted grids close to the wall. As a result, wall functions are not necessary, not even at full scale. More details about the solution strategy can be found in [3].

PARNASSOS makes use of a surface-fitting method to determine the free surface, based on the steady iterative formulation which, contrary to almost all other RANS/FS methods, involves no time-dependent terms. The problem is solved by an iterative procedure, instead of by time integration. This iteration is based on the use of a combined free-surface condition that is obtained by substituting the wave elevation from the dynamic boundary condition into
the kinematic boundary condition for the free surface. Together with the dynamic condition it describes exactly the same problem as the original set of conditions.

For the analysis of the flow past the propeller, use is made of a boundary element method (BEM) that solves the incompressible potential flow equations for lifting and non-lifting bodies. The method, designated PROCAL, is being developed within MARIN’s Cooperative Research Ships (CRS) for the unsteady analysis of cavitating propellers operating in a prescribed ship wake. It has been validated for open water characteristics, shaft forces, sheet cavitation inception and extent and hull pressure fluctuations. The code is a low order BEM that solves for the velocity disturbance potential. Initial validation studies and details on the mathematical and numerical model can be found in [6,7].

The steady RANS solver and the unsteady boundary element method are coupled in the following way:

- First a RANS computation is made for the case without propeller. This provides the resistance and the nominal wake field at the propeller plane.
- Then, a first propeller computation is made using the BEM, for the propeller in this wake field at the prescribed rotation rate. This provides a thrust and loading distribution. The unsteady loading distribution, in a ship-fixed coordinate system, is averaged in time for all blade positions to produce a steady, but axially, circumferentially and radially non-uniform force distribution. This is interpolated to the RANS grid.
- We restart the viscous-flow computation from the previous solution and impose this loading distribution as a force field acting on the flow. This yields a new total wake field, from which we then subtract the induced velocities coming from the BEM to obtain the first estimate of the effective wake field.
- We iterate between both methods until the thrust and torque coefficients and the effective inflow to the propeller do not change any more.

4 COMPUTATIONAL DOMAIN AND GRID GENERATION

The computational domain extended from the inflow boundary, located 1Lpp in front of the bow, to the outflow boundary, 1.5Lpp behind the transom. The lateral outer boundary is a quarter of a cylinder with axis y=z=0 and radius 1Lpp. At this boundary tangential velocities and pressure found from a potential-flow computation are imposed. In the resistance computations only the starboard side of the ship was taken into account, due to symmetry considerations. This mesh consisted of 512 x 200 x 52 cells in the streamwise, girthwise and wall-normal directions, respectively. Approximately 200 cells in the streamwise direction were distributed along the hull. The cells in the wall-normal direction were contracted strongly towards the hull, to capture the gradients in the boundary layer, resulting in y+ values below 0.4 in all computations. For the power predictions the mesh was mirrored to port side as well, resulting in a mesh with 10.6M cells.

5 RESISTANCE PREDICTIONS

Resistance computations have been performed at seven different speeds. Three of those,
namely 8, 10 and 12 knots, were three of the cases that were requested for the resistance predictions at the workshop. Two more speeds, namely 9 and 13 knots, resulted from the power predictions at the workshop. These values served as indications of the ship speeds that were obtained from the trials at the given RPM’s. During the workshop the actually obtained ship speeds were revealed and for two of those, namely 9.25 and 11.58 knots, resistance and power predictions have been performed afterwards as well.

Convergence levels that could be obtained in the resistance computations turned out to be speed dependent. At the lower speeds, 8, 9 and 9.25 knots, but also at 13 knots both the RANS-equations and the free-surface updates in our steady iterative approach were well converged. Maximum changes in the pressure coefficients and the dimensionless velocities at any point in the domain had dropped below $1 \times 10^{-5}$; maximum changes in the free surface elevation were less than 1 mm. For these cases any significant influence of the iterative errors on the solutions can safely be disregarded. For the remaining three speeds, 10, 11.58 and 12 knots, convergence was somewhat more cumbersome to achieve. Maximum changes in the solutions were typically 1-2 orders of magnitude larger than for the other speeds. The largest changes in the solutions consistently occurred near the free surface, either near the bow or near the rudder. As will be discussed in more detail later, the solutions showed clear indications of the occurrence of an overturning bow wave at most of the speed range, while as a consequence of the loading conditions during the trials the rudder protruded through the free surface at all speeds. The difficulties in the convergence behavior are likely to be related to these features. Table 2 presents the main flow parameters for the resistance computations.

<table>
<thead>
<tr>
<th>Knots</th>
<th>Reynolds number (Rn)</th>
<th>Froude number (Fn)</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>4.116</td>
<td>6.43 x 10^8</td>
</tr>
<tr>
<td>9</td>
<td>4.630</td>
<td>7.23 x 10^8</td>
</tr>
<tr>
<td>9.25</td>
<td>4.759</td>
<td>7.43 x 10^8</td>
</tr>
<tr>
<td>10</td>
<td>5.144</td>
<td>8.03 x 10^8</td>
</tr>
<tr>
<td>11.58</td>
<td>5.957</td>
<td>9.30 x 10^8</td>
</tr>
<tr>
<td>12</td>
<td>6.173</td>
<td>9.64 x 10^8</td>
</tr>
<tr>
<td>13</td>
<td>6.688</td>
<td>1.04 x 10^9</td>
</tr>
</tbody>
</table>

Figure 1 presents the resistance values that have been obtained. The red diamonds correspond to the CFD predictions, containing the contribution of the hull and the rudder only. The blue squares are the same values augmented with a correlation allowance, $C_A$, to account for the features that have not been taken into account in the simulations. Finally the dashed line corresponds to an estimated resistance curve for the present test case that is based on MARIN’s statistical prediction method (DESP).

At lower speeds the computations show a smooth increase in the resistance with increasing speed, practically parallel to the DESP curve. At higher speeds the behavior of the predicted resistance is more irregular. Compared to the DESP curve the resistance at 11.58 knots seems to be somewhat high, while the opposite holds for the two highest speeds. Recalling that the convergence levels of the computations in the range between 10 and 12 knots were the least, one might be tempted to fit the resistance curve through the three lowest and the highest point.
That then would lead to a curve that runs slightly less steep than the one from DESP. However, it has to be emphasized that the comparison with the statistical method can only be used as indicative, not as a validation. Further confidence in the resistance predictions thus has to come from mutual comparison of numerical results. When doing so for the results that were provided at the workshop, however, no less than 11-16 per cent spreading was reported between the resistance values, after disregarding the highest and lowest values. This indicates that the full-scale CFD resistance predictions are not yet sufficiently mutually consistent, and there is work to be done.

![Figure 1: The resistance curve.](image)

As mentioned above, the CFD predictions (the red diamonds in the figure) contain the contributions of the hull and the rudder only. Added to that should be resistance estimations of firstly the bilge keels and anodes that prior to the workshop were removed from the hull geometry after the 3D laser scan. This was done because the quality of these scanned surfaces was judged to be insufficient. Their contributions may be a few per cent of the bare-hull resistance. Secondly the resistance of the superstructure is needed. That might be included in the simulation, or determined in a separate simulation for the superstructure only, or estimated using the expression by Bowden-Davison, as recommended by the ITTC. In the proceedings of the workshop an added air resistance of up to seven per cent is reported. Thirdly at full scale the resistance increase due to surface roughness cannot be ignored. This can be included through an empirical relation of e.g. Townsin, or numerically through the adaptation of the boundary conditions in the turbulence model. As additional input these two methods then require the specification of the mean apparent amplitude (MAA) or an equivalent sand grain roughness height. For the present ship the measured MAA, after cleaning the hull, was reported to vary over the hull in the range 70-270 μm. At the design speed, according to Townsin’s formula, this would add something between 2-8 per cent to the bare-hull resistance.

These, and other contributions that are not further discussed here, easily add 10-20 per cent to the bare-hull resistance. At MARIN all the contributions mentioned above are typically
collected into a single correlation allowance that is added to the full-scale bare-hull resistance. For the present ship this allowance has been determined as $C_A = 0.456 \times 10^{-3}$. As the predicted resistance coefficients lie in the range $2.37 - 2.75 \times 10^{-3}$, this indeed corresponds to a resistance increase of 15-20 per cent (the blue squares in the resistance curve). These values will be used later on in this paper to evaluate the self-propulsion points.

Apart from the comparison of the total resistance, it can be very worthwhile to compare the predicted frictional resistance coefficients with well-known friction lines, such as for instance the ITTC-’57 model-ship correlation line, the Grigson line or the more recently derived numerical friction lines [8]. As the displacement effect of the ship results in an increase of the velocity along most of the ship, it can be expected that the frictional resistance of the ship is higher than the frictional resistance of an equivalent flat plate. How much exactly is not known, but values close to, or even below, the friction lines can be regarded as indications of numerical errors. Indeed, such values occurred at the present workshop, but also at the 2015 Tokyo Workshop. In the discussions following that workshop, it was mentioned that one of the likely causes of under-estimated friction is the (improper) use of wall functions. For our present results, where no wall functions have been used, Figure 2 compares the predicted friction coefficients with several friction lines. The dash-dotted black line in the figure corresponds to an eight per cent upward shift of the numerical friction lines corresponding to Menter’s one-equation turbulence model, which is the model used in these simulations. Thus we find a friction coefficient that is consistently 8 per cent above that line, and that over the entire speed range. Obviously, a different percentage is found when comparing to another friction line.

![Figure 2: Comparison of the predicted friction coefficients with several friction lines.](image)

We now consider some features that characterize the wave pattern and flow field around the ship. Figure 3 illustrates the predicted bow wave system at various speeds, colored with the axial velocity component at the free surface. With increasing speed a pronounced bow wave develops, with the highest wave crest located at some distance in front of the ship. With increasing wave height, and thus with increasing ship speed, the axial velocity in the top of the wave crest decreases; from $u/V_s = 0.22$ at 9.25 knots, to $u/V_s = 0.09$ at 11.58 knots and at 12 knots the axial velocity has become negative. This, together with the steepness of the bow
wave, is a clear indication that the bow wave will be over-turning in reality. It is evident that an over-turning bow wave cannot be a solution in the surface-fitting method that has been implemented in PARNASSOS. And this may also explain why good convergence was found to be more difficult to achieve in this region at the higher speeds. For comparison the bottom-right of Figure 3 shows a result obtained with ReFRESCO (http://www.refresco.org), which uses a volume-of-fluid (VOF) method to model the free surface. Shown is an iso-surface through the points where the air volume fraction equals one half. Here the occurrence of an over-turning bow wave in reality is even more convincing.

Figure 3: Development of the bow-wave system.

Figure 4 shows the axial velocity field near the stern for the lowest and the highest speeds considered here. Due to the specific loading condition during the trials (\(T_f = 4.988\) m., \(T_a = 5.597\) m. compared with a design draught of \(T = 8.5\) m.) the stern and the transom of the ship are lifted well above the free surface, also when sailing. At all but the highest speed, the waterline ends just above the ship’s gondola. The top part of the rudder is above the water surface over the entire speed range. Consequently the rudder creates its own wave pattern with diverging waves coming from the leading and the trailing edge. At lower speeds the trailing-edge wave was found to be quite short with low velocities in the top, possibly given rise to the formation of a short spilling breaker in reality. At higher speeds (higher Froude numbers) the top of this wave shifted aft and became less steep.

It can also be seen from Figure 4 that there will be not much clearance between the tip of the propeller blades and the free surface at the present loading condition.

To finish the discussion of the resistance computations Figure 5 presents the nominal wake
fields at the propeller plane for the 8 and 13 knots cases. Both fields can be characterized by the presence of two longitudinal vortices in the bottom half of the discs. Note that at the design condition these vortices are most likely located in the top half of the disc, where they are used to make the axial velocity field more homogeneous. But under the present loading condition they have shifted downwards. The axial velocity field at 8 knots shows somewhat higher velocities in the top of the disc and below the shaft than at 13 knots, but for the rest there is not much difference between them. The nominal wake fractions integrated over these discs have been found to change from $1 - w_{\text{nom}} = 0.52$ at 8 knots to $1 - w_{\text{nom}} = 0.49$ at 13 knots.

![Figure 4: Detail of the flow fields near the stern.](image)

![Figure 5: The nominal wake fields at 8 and 13 knots.](image)

6 PROPULSION PREDICTIONS

For the self-propulsion predictions at the workshop it was requested to determine the ship speeds that were obtained given the shaft speeds recorded during the trials, namely 71.6, 91.1 and 106.4 RPM. However, such an approach would require an iterative adjustment of the inlet velocity in the RANS computations, which can be computationally expensive. The approach adopted by most participants at the workshop, and also by us, was to perform a number of computations at fixed ship speed and RPM, and determine the self-propulsion point from those. Indications of the obtained ship speeds at the RPM’s mentioned above were provided prior to the workshop as 9, 12 and 13 knots, respectively. Here we will report our results for the two lowest shaft speeds. For efficiency reasons the approach we have adopted for these computations is as follows:
First we restarted the RANS free-surface computations from the resistance results, imposing the axial loading distribution from the propeller only (after averaging). In such a way port-starboard symmetry was preserved and the first iterations for the propeller-hull interaction could efficiently be performed for one side of the ship only.

Secondly, we mirrored the solution from starboard to port side and restarted the computations including the radial and tangential loading of the propeller as well. Thus the asymmetry of the axial loading and the influence of the propeller swirl on the flow field were taken into account.

Our contributions to the workshop included the cases with the shaft speeds of 71.6 and 91.1 RPM. The corresponding simulation were performed at the indicated ship speeds of 9 and 12 knots, respectively. During the workshop the actually achieved ship speeds from the trial results were given, being 9.25 knots at 71.6 RPM and 11.58 knots at 91.1 RPM. Computations using these conditions have been performed after the workshop, and results of all four simulations will be discussed here.

Table 3: Main results for the analysis of the power predictions.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>1.193</td>
<td>109.44</td>
<td>150.61</td>
<td>174.3</td>
<td>109.3</td>
<td>0.166</td>
<td>0.200</td>
<td>819.8</td>
<td>0.549</td>
<td>0.236</td>
</tr>
<tr>
<td>9.25</td>
<td>1.193</td>
<td>116.07</td>
<td>157.82</td>
<td>170.7</td>
<td>107.6</td>
<td>0.162</td>
<td>0.197</td>
<td>807.3</td>
<td>0.546</td>
<td>0.245</td>
</tr>
<tr>
<td>11.58</td>
<td>1.518</td>
<td>211.10</td>
<td>258.98</td>
<td>270.1</td>
<td>169.6</td>
<td>0.159</td>
<td>0.192</td>
<td>1618</td>
<td>0.552</td>
<td>0.177</td>
</tr>
<tr>
<td>12</td>
<td>1.518</td>
<td>216.76</td>
<td>273.50</td>
<td>268.5</td>
<td>168.7</td>
<td>0.158</td>
<td>0.191</td>
<td>1609</td>
<td>0.535</td>
<td>0.211</td>
</tr>
</tbody>
</table>

Table 3 lists the main parameters that are required to analyze the power predictions, augmented with the corresponding resistance values taken from the previous section and the thrust deduction fraction. First consider the imbalance between the thrust, T, and the resistance force acting on the hull in propulsion plus the correlation allowance, Rₜ + Cₐ. At 9 knots we found a higher thrust than resistance, which would lead to an acceleration of the ship. At 12 knots the predicted thrust was somewhat lower than the resistance, and thus the ship would decelerate. The trial data indicate that the ship speed at 71.6 RPM is indeed higher than 9 knots, namely 9.25 knots, and at 91.1 RPM lower than 12 knots, namely 11.58 knots. However, also in the new computations that have been performed at exactly those conditions, we do not find a balance between thrust and resistance; in both cases the thrust exceeds the resistance by 11-13kN, indicating an increase of the ship speed compared to the trial data. To determine the magnitude of the ship speed at self-propulsion point we use the open-water diagram of the propeller, shown at the left-hand side of Figure 6. This diagram gives a relation between the thrust coefficient and the advance coefficient J. If it is assumed that the effective wake fraction is constant than J = (1-wₑff)Vₖ/(nD) will vary linearly with the ship speed, which for a small difference in speed is a reasonable assumption. From an estimated increase in speed, a corresponding decrease of the thrust can then be calculated. An estimation for the new value of the resistance that has to be overcome can then be obtained from the resistance at the higher speed (from the resistance curve in Figure 1) plus the thrust deduction times the new thrust.

For example, at 9.25 knots we found a thrust of T = 170.7kN and a thrust coefficient Kₜ =
0.162. In the open-water diagram this corresponds to an advance coefficient of $J=0.4062$. We estimate an increased ship speed as $V_s^*=9.53$ knots. This gives a new advance coefficient of $J = (V_s^*/V_s) \times J^*=0.4185$. From the open-water diagram we read a new thrust coefficient $K_T^*=0.156$. Thus the thrust changes to $T^*= (K_T^*/K_T) \times T = 165.2kN$. From the resistance curve we read the resistance at the new speed, $R^* = 125.3kN$. The total resistance force that has to be overcome by the thrust then equals $R_T^* = R^* + tT^* = 165.8kN$. And now $R_T^*$ and $T^*$ are practically equal. Thus to obtain self-propulsion in the present computations at 71.6 RPM, the ship speed should be increased to 9.53 knots, 0.28 knots higher than determined from the trials. The delivered power at that speed can then be determined as $P_D = 765.5kW$. Performing a similar analysis for the 11.58 knots case results in an increase of the ship speed with 0.17 knots to 11.75 knots, at a delivered power of $P_D = 1570kW$.

A comparison of these points with the trial data is given at the right-hand side of Figure 6.

![Figure 6: The open-water diagram (left) and the speed-power curve (right).](image1)

![Figure 7: Detail of the flow fields near the stern at the starboard part of the domain.](image2)

The results that have been submitted to the workshop are indicated by red, open squares, and the new results by green, solid squares. For the lowest RPM both computations (at 9 and 9.25 knots) result in the same estimated speed and power at self-propulsion, as was to be expected. At the higher RPM, however, the computations at 11.58 and 12 knots do not result in the same self-propulsion point, indicating that there must be some inconsistencies between these computations. From Table 3 it can for instance be read that the difference in the effective wake fraction and the thrust deduction is much larger for these two cases, compared to the 9 and 9.25 knots cases. Finally from Figure 7 it can be seen that the power predictions
are also complicated by the flow field. At the left-hand side (9.25 knots case) a clear region with reversed flow (indicated by black contour flooding) at the free surface is present above the propeller in the solution. At 11.58 knots, the right-hand side of the figure, this region has extended upstream to the stern.

6 DISCUSSION

Figure 8 shows the speed-power predictions of all participants of the workshop, reproduced from [2] with kind permission of the organizers. For each of the three cases the spreading of the predicted ship speed at given RPM is more than 1 knot, indicating that in general these predictions are not yet sufficiently accurate, or well-controlled. Evidently, numerous error sources can be the cause of the observed spreading: discretization errors due to insufficient grid resolution or time stepping, iterative errors due to lack of convergence, influence of turbulence models on the predicted resistance and propeller inflow, the influence of wall functions, etc. In the absence of any experimental data on e.g. the resistance or the wake field, this spreading can only be reduced by careful analysis of the numerical results and mutual comparison of numerical results.

However, it can also be that the present test case gives a somewhat negative view on the capability of CFD to predict power and RPM at full scale. We have illustrated in our paper that due to the specific loading conditions at the trials, flow features occur that complicate the CFD predictions, such as bow-wave breaking and flow reversal at the free surface just above the propeller. These are less likely to occur at the design draught of the ship, so it may well be that the accuracy of the predictions and the consistency of the results increase if the same exercise would be performed at the design conditions. But then we are obviously lacking any trial data for validation.

On the other hand, looking at the figure it can also be seen that the majority of the CFD results are consistently located below the experimental speed-power curve, resulting in an over-prediction of the ship speed compared to the trials. Or, equivalently, one could claim that
the resistance of the ship is consistently under-estimated. Realizing that the correlation allowance that has to be added to the predicted resistance of the ship is in general determined for newly-built ships, it might also be that the resistance of this ship, having served for over twenty years, is higher than expected. If that would be true, most CFD results would be in better agreement with the trial data.

6 CONCLUSIONS

Using a hybrid RANS-BEM method we have over-estimated the ship speed at two self-propulsion points by 0.17-0.28 knots, compared to the trial data. Although that is judged to be somewhat too high to be acceptable in practical design projects, the CFD was complicated by the flow features caused by the specific loading condition at the trials, which are not likely to occur at the design conditions.

The frictional resistance of the ship exceeded the value of the corresponding numerical friction line in our computations by 8 per cent, consistently over the entire speed range (8-13 knots).

There is a significant spreading in the results of all participants. A detailed evaluation and mutual comparison of all aspects in the resistance and propulsion computations is required to reduce this spreading to more acceptable levels in future workshops.

ACKNOWLEDGEMENT

This research is partially funded by the Dutch Ministry of Economic Affairs.

REFERENCES

SHIP RESISTANCE PREDICTION: VERIFICATION AND VALIDATION EXERCISE ON UNSTRUCTURED GRIDS

P. CREPIER*

*Maritime Research Institute Netherlands (MARIN)
P.O. Box 28
6700 AA Wageningen, The Netherlands
e-mail: p.crepier@marin.nl, web page: http://www.marin.nl

Key words: CFD, KVLCC2, Verification, Validation, Double-body, Unstructured grid

Abstract. The prediction of the resistance of a ship is, together with the propeller performance prediction, part of the key aspects during the design process of a ship, as it partly ensures the quality of the power-prediction. Body fitted structured grids for ship simulations can be rather challenging and time consuming to build, especially when dealing with appended ship geometries. For this reason, unstructured hexahedral trimmed grids are more and more used. Such grids can be build by various CFD package such as CD-Adapcos Star CCM+, NUMECA’s Hexpress grid generator or OpenFOAMS’s SnappyHexMesh. Although their use is increasing or even already adopted, the numerical uncertainty of these simulations seems to be a well-kept secret.

In the study presented, an attempt at quantifying the numerical uncertainty of the resistance for the combination of the RANS Solver ReFRESCO [1] with grids generated using the commercial package Hexpress is made. The studied case is the flow around the bare-hull KVLCC2 at model scale Reynolds number. Extensive verification and validation on the same test case has already been published for the combination of ReFRESCO and structured grids by Pereira et al. [2].

The method to generate grids as geometrically similar as possible is presented, and the uncertainty analysis by L. Eça and M. Hoekstra [3] is performed on the integral results obtained. The simulations are performed using the $k-\omega$ SST, $k-\omega$ TNT and the $k-\sqrt{k}L$ turbulence models. The velocity fields calculated in the propeller plane are compared to the measured ones and to the results obtained by Pereira et al. [2] on structured grids.

The results show that the differences with the experimental results are in the same range as the differences obtained with structured grids. The numerical uncertainties are, however, higher. They are also strongly dependent on the turbulence model used, like for structured grids, and are spread between 1.3% and 12%.

Concerning the wake flow details, not all features present in the experimental results are obtained and, compared to structured grids, the flow features are smoothed. The wake flow is also influenced by the turbulence modelling and needs to be addressed in more detail.
1 INTRODUCTION

Ship resistance predictions by means of Computational Fluid Dynamic (CFD) simulations is progressively taking over ship model testing, especially in the early design stages of the design loop. This sort of calculations has nearly become daily routine, but the accuracy of the results is often overlooked. While a lot of effort is spend during workshops, like the Gothenburg or Tokyo workshops, to gather validation material for various types of calm-water flows, not so much publications about verification of the simulations performed is available. Verification and Validation are two entirely different exercises as explained by Roache [4]: Verification is a mathematical exercise that aims at showing that we are solving the equations right, and validation is an engineering exercise to show that we are solving the right equations.

For ship flow simulation it is common to use body-fitted hexahedral trimmed meshes because they are easy to set-up even for complex geometries like appended ships. Such grids can be built by most of the popular CFD software package like CD-Adapco’s Star CCM+, NUMECA’s Hexpress or OpenFOAMS’s SnappyHexMesh.

L. Eça and M. Hoekstra [3] proposed a method to estimate the numerical uncertainty of numerical simulations based on grid refinement studies of geometrically similar grids. Generating the appropriate sets of grids is straightforward when using structured grids but it becomes more challenging when working with unstructured meshes. This is most likely one of the main reasons for the lack of verification studies, in addition to being a rather costly exercise.

In the present study, the point of interest is the flow around the KVLCC2 at model scale for which plenty of data is available. The grid sets are built using NUMECAs grid generator Hexpress. In section 2 a summary of the test case and in-depth details about the method used to generate grids which are as geometrically similar as the grid generator allows. The details about the RANS solver and numerical settings are provided in section 3. In section 4, the obtained results are detailed in terms of numerical convergence, and the uncertainty analysis is performed on the resistance components. The details of the wake flow are also shown. These results are also compared to those obtained by Pereira et al.[2] for the same test case but with structured grids. Finally, in section 5, the conclusions of the findings are summarised.

2 GEOMETRY AND GRID GENERATION METHOD

2.1 KVLCC2

The object of the present study is the KVLCC2. A summary of its main particular, scale ratio and Reynolds number are provided in table 1 and a side view of the vessel is shown in figure 1.

Figure 1: Side view of the KVLCC2
Table 1: Main particulars of the KVLCC2

<table>
<thead>
<tr>
<th>Particular</th>
<th>Symbol</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length between perpendiculars</td>
<td>$L_{pp}$</td>
<td>320.0</td>
<td>[m]</td>
</tr>
<tr>
<td>Width</td>
<td>$B$</td>
<td>58.0</td>
<td>[m]</td>
</tr>
<tr>
<td>Draught</td>
<td>$T$</td>
<td>10.8</td>
<td>[m]</td>
</tr>
<tr>
<td>Scale</td>
<td>$\lambda$</td>
<td>58.0</td>
<td>[-]</td>
</tr>
<tr>
<td>Froude Number</td>
<td>$F_r$</td>
<td>0.142</td>
<td>[-]</td>
</tr>
<tr>
<td>Reynolds Number</td>
<td>$R_e$</td>
<td>$5.80 \times 10^6$</td>
<td>[-]</td>
</tr>
</tbody>
</table>

2.2 Isotropic volume grid generation

The grids used in this study are so called trimmed meshes. In these meshes, a background grid with large cells is defined and then, the cells intersecting the input geometry are successively divided into 8 smaller cells to adapt to the details of the geometry. The main user input is the cell size for the initial grid, the refinement degree for each geometrical feature that should be captured, and the size of the transition zone between two refinement levels called diffusion depth $d$. Once a sufficient resolution is obtained at the places of interest, an anisotropic sub-layer of cells can be inserted to provide a grid suited to properly capture the boundary-layer on the walls present in the grid. The grid sets built for this study are based on an initial coarse grid which is successively refined to obtain, in total, five grids. To obtain grids that are as geometrically similar as possible the following method is used:

1. The initial cell size is decreased by a factor 2, 3, 4 and 5 in each direction by using 2, 3, 4 or 5 times more cells in each direction.
2. The surface refinement degree is kept constant throughout the sets: if, for instance, 6 refinements levels are set in the initial coarse grid, the same 6 successive refinements are performed for the other grids.
3. The size of the transition region, so-called diffusion depth $d$, is adapted such that it matches the expected final size of the grid. Details of the values used to generate the grids used in this study are provided in section 2.4.
4. The anisotropic sub-layer settings are adapted to account for the refinement performed. This step is detailed in paragraph 2.3.

Examples of the volume obtained grid after the third step are shown in figure 2 for a simple case.
2.3 Anisotropic sub-layer grid generation

The size of the cells inserted in the anisotropic sub-layer follows a geometric series of first term $S_0$, corresponding to the first cell size, and ratio $r$. The size of the $n^{th}$ cell is then defined as follow:

$$S_n = S_0 r^n$$  \hspace{1cm} (1)

With such a definition, dividing the initial cell size, and keeping the ratio constant in all grids will not result in geometrically similar meshes. As shown in figure 3(a), when dividing the first cell size by two and keeping the ratio constant, between 13 and 14 cells are required to obtain a distance covered by 10 cells with the initial settings instead of 20.

To obtain geometrically similar grids, both the first cell size and and ratio should be adapted following Equations 2 and 3:

$$S_n = S_0 \frac{1 - r_1^n}{1 - r_1} \hspace{1cm} (2)$$

$$r_n = r_1^n \hspace{1cm} (3)$$

Where $S_0$ and $r_1$ are respectively the first cell size and growth ratio in the initial coarse grid, $S_n$ and $r_n$ the first cell size and growth ratio for the grid refinement $n$, $n = 1$ corresponding the coarsest grid.

Using the example in Figure 3(a) and setting up the geometric series properly, twice more cells are required with a refinement of 2 and 3 times more cells with a refinement 3, as shown in Figure 3(b).

Figure 3: Anisotropic sub-layer refinement set-up
2.4 Grid sets

Following the method described, two grid sets have been built. The sets differ only by the first cell size which is smaller in the second set, the isotropic volume grids are identical in both sets. All the grids built are 6 ship lengths long (3 astern, 2 ahead) and 2 ship lengths wide and deep. The ship geometry is split in 5 different parts, aft-ship, mid-ship, bilge, fore-ship and bulbous bow, to properly set-up the surface refinements. A box of volume refinement is used around the whole ship to keep the grid density reasonable near the ship.

Views of the CFD domain, surfaces and boxes defined around the ship are shown in figure 4. In table 2, details of the settings used for each surface patch and the box are provided.

![Image of CFD domain and definition of the surfaces and box around the ship]

Figure 4: CFD domain and definition of the surfaces and box around the ship

<table>
<thead>
<tr>
<th>Part</th>
<th>Aft-ship</th>
<th>Mid-ship</th>
<th>Bilge</th>
<th>Fore-ship</th>
<th>Bulbous Bow</th>
<th>Box ship</th>
</tr>
</thead>
<tbody>
<tr>
<td>Level</td>
<td>7</td>
<td>6</td>
<td>7</td>
<td>7</td>
<td>8</td>
<td>6</td>
</tr>
</tbody>
</table>

Table 2: Refinement levels set for each part of the grid

As detailed in section 2.2, to perform the refinement of the isotropic volume grid, only the number of cells in the initial grid and the transition layer are adapted. Details of the settings used to build the 5 grids per set are detailed in table 4. The growth ratio in the anisotropic sub-layer, which starts at 1.2 in the coarsest grids, is also adapted accordingly to the refinement level of each grids. The number of cells obtained in each grid, as well as the average $Y^+$ obtained with the $k-\omega$ SST turbulence model, are presented in table 3.

3 RANS SOLVER AND NUMERICAL SETTINGS

The simulations are performed using the URANS (Unsteady Reynolds Average Navier Stokes) CFD code ReFRESCO [1]. The QUICK scheme is used for the discretisation of the convective flux in the momentum equations. Three different turbulence models are used in this study, namely the $k-\omega$ SST [5], $k-\omega$ TNT [6] and the $k-\sqrt{k}L$ [7]. Upwind is used for the convective flux discretisation in the turbulence equations.
Table 3: Total number of cells in each grid in million and average $y^+$

<table>
<thead>
<tr>
<th>Grid set</th>
<th>1 Cell count</th>
<th>$\overline{y^+}$</th>
<th>2 Cell count</th>
<th>$\overline{y^+}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid 1</td>
<td>0.405</td>
<td>0.60</td>
<td>0.541</td>
<td>0.062</td>
</tr>
<tr>
<td>Grid 2</td>
<td>2.79</td>
<td>0.30</td>
<td>3.826</td>
<td>0.030</td>
</tr>
<tr>
<td>Grid 3</td>
<td>8.53</td>
<td>0.19</td>
<td>11.9</td>
<td>0.020</td>
</tr>
<tr>
<td>Grid 4</td>
<td>19.1</td>
<td>0.14</td>
<td>27.1</td>
<td>0.015</td>
</tr>
<tr>
<td>Grid 5</td>
<td>35.8</td>
<td>0.12</td>
<td>51.3</td>
<td>0.012</td>
</tr>
</tbody>
</table>

Table 4: Number of cells in the three directions and diffusion depth values used for the grid sets

<table>
<thead>
<tr>
<th>Grid</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nx</td>
<td>12</td>
<td>24</td>
<td>36</td>
<td>48</td>
<td>60</td>
</tr>
<tr>
<td>Ny</td>
<td>4</td>
<td>8</td>
<td>12</td>
<td>16</td>
<td>20</td>
</tr>
<tr>
<td>Nz</td>
<td>4</td>
<td>8</td>
<td>12</td>
<td>16</td>
<td>20</td>
</tr>
<tr>
<td>d</td>
<td>1</td>
<td>3</td>
<td>5</td>
<td>7</td>
<td>9</td>
</tr>
</tbody>
</table>

As boundary condition of the problem, an inflow condition is imposed at the plane upstream of the ship and an outflow (Neumann) at the plane downstream. Symmetry conditions are imposed at the symmetry plane of the ship and at the top boundary. A constant pressure is imposed at the bottom and far-field left side of the ship. On the ship itself, a no-slip condition is used as the grid sets built are contracted enough towards the wall such that no wall functions are used.

4 RESULTS

4.1 NUMERICAL CONVERGENCE

In most of the simulations performed, all residuals are, on average, converged below $10^{-6}$ except for a few simulations where usually one of the turbulence quantities stagnates at a higher level of $10^{-4}$. This is especially true for the simulation with the $k-\omega$ based model where the $\omega$ becomes more difficult to solve when the grid is refined.

The results show that the residuals of the continuity and momentum equations are highest around the propeller plane while the turbulence residuals are highest at the bow of the ship. Figure 5(a) shows the convergence history of the root mean square residual for a case converging properly, and figure 5(b) for a stagnating case. Both figures are for the $k-\omega$ SST turbulence model.
4.2 Forces uncertainty

A direct result of the simulations is the resistance of the ship. The total forces obtained and its components, pressure and friction, are normalized using equation 4:

\[ C_i = \frac{F_i}{\frac{1}{2} \rho U^2_{\infty} S} \]  

(4)

Where \( i \) is the component of the force, either pressure part, friction part or total, \( \rho \) is the fluid density, \( U^2_{\infty} \) is the free-stream velocity and \( S \) is the wetted surface of the ship.

The results for the three turbulence models are gathered in table 5 for the first grid set and in table 6 for the second one. In table 7, the experimental results as well as the results obtained by Pereira et al. [2] for structured grids with his finest grid are listed.

For the simulations performed with \( k - \omega \) SST, the pressure drag decreases when the grid is refined while the friction drag increases. Both the friction and pressure drag increase when the wall resolution increases. The same behavior is obtained with \( k - \omega \) TNT.

For the \( k - \sqrt{kL} \), the pressure coefficient shows this similar behavior but the friction drag is highest for the intermediate grids 2 and 3 and decreases slightly for the finest grid.

When comparing the total drag, on average, all simulations underestimate the experimental value. The maximum difference obtained with experimental drag is around 3\% for \( k - \omega \) TNT, 1.5\% for \( k - \omega \) TNT and 4\% for \( k - \sqrt{kL} \).

The uncertainty analysis proposed by L. Eça and M. Hoekstra [3] has been performed in order to quantify the discretisation error obtained with these grids. The extrapolation is performed using only the four finest grids of each set, meaning that the grid 1 of each set is not used. For this reason, the uncertainty for grid 1 is not given. The obtained value is only used to show the resulting trend when using coarser grids.
The obtained uncertainties are gathered in table 8 for the first grid set and in table 9 for the second one. For comparison, the uncertainty obtained by Pereira et al. with his finest grid are given at the bottom of each tables.

**Table 5:** $C_p$, $C_f$ and $C_t$ obtained for the first grid set

<table>
<thead>
<tr>
<th>Grid</th>
<th>$k - \omega$ SST</th>
<th>$k - \omega$ TNT</th>
<th>$k - \sqrt{kL}$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$C_p$</td>
<td>$C_f$</td>
<td>$C_t$</td>
</tr>
<tr>
<td>Grid 1</td>
<td>0.78</td>
<td>3.22</td>
<td>4.00</td>
</tr>
<tr>
<td>Grid 2</td>
<td>0.64</td>
<td>3.34</td>
<td>3.98</td>
</tr>
<tr>
<td>Grid 3</td>
<td>0.62</td>
<td>3.37</td>
<td>3.99</td>
</tr>
<tr>
<td>Grid 4</td>
<td>0.62</td>
<td>3.38</td>
<td>4.00</td>
</tr>
<tr>
<td>Grid 5</td>
<td>0.63</td>
<td>3.38</td>
<td>4.01</td>
</tr>
</tbody>
</table>

**Table 6:** $C_p$, $C_f$ and $C_t$ obtained for the second grid set

<table>
<thead>
<tr>
<th>Grid</th>
<th>$k - \omega$ SST</th>
<th>$k - \omega$ TNT</th>
<th>$k - \sqrt{kL}$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$C_p$</td>
<td>$C_f$</td>
<td>$C_t$</td>
</tr>
<tr>
<td>Grid 1</td>
<td>0.80</td>
<td>3.38</td>
<td>4.19</td>
</tr>
<tr>
<td>Grid 2</td>
<td>0.65</td>
<td>3.43</td>
<td>4.08</td>
</tr>
<tr>
<td>Grid 3</td>
<td>0.63</td>
<td>3.43</td>
<td>4.05</td>
</tr>
<tr>
<td>Grid 4</td>
<td>0.62</td>
<td>3.43</td>
<td>4.05</td>
</tr>
<tr>
<td>Grid 5</td>
<td>0.64</td>
<td>3.42</td>
<td>4.05</td>
</tr>
</tbody>
</table>

**Table 7:** $C_p$, $C_f$ and $C_t$ obtained by Pereira et al. and $C_t$ obtained experimentaly (EFD)

<table>
<thead>
<tr>
<th>Case</th>
<th>$C_p$</th>
<th>$C_f$</th>
<th>$C_t$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pereira et al. $k - \omega$ SST</td>
<td>0.68</td>
<td>3.38</td>
<td>4.06</td>
</tr>
<tr>
<td>Pereira et al. $k - \omega$ TNT</td>
<td>0.66</td>
<td>3.47</td>
<td>4.13</td>
</tr>
<tr>
<td>Pereira et al. $k - \sqrt{kL}$</td>
<td>0.66</td>
<td>3.33</td>
<td>3.98</td>
</tr>
<tr>
<td>EFD</td>
<td>-</td>
<td>-</td>
<td>4.11</td>
</tr>
</tbody>
</table>
Table 8: Uncertainties, in percent, obtained for $C_p$, $C_f$ and $C_t$ with the first grid set

<table>
<thead>
<tr>
<th>Grid</th>
<th>$k-\omega$ SST</th>
<th>$k-\omega$ TNT</th>
<th>$k-\sqrt{kL}$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$C_p$</td>
<td>$C_f$</td>
<td>$C_t$</td>
</tr>
<tr>
<td>Grid 2</td>
<td>26.7</td>
<td>5.0</td>
<td>4.1</td>
</tr>
<tr>
<td>Grid 3</td>
<td>38.8</td>
<td>2.5</td>
<td>2.9</td>
</tr>
<tr>
<td>Grid 4</td>
<td>38.0</td>
<td>1.4</td>
<td>2.4</td>
</tr>
<tr>
<td>Grid 5</td>
<td>34.0</td>
<td>1.0</td>
<td>2.0</td>
</tr>
<tr>
<td>Pereira et al.</td>
<td>0.95</td>
<td>1.7</td>
<td>1.5</td>
</tr>
</tbody>
</table>

Table 9: Uncertainties, in percent, obtained for $C_p$, $C_f$ and $C_t$ with the second grid set

<table>
<thead>
<tr>
<th>Grid</th>
<th>$k-\omega$ SST</th>
<th>$k-\omega$ TNT</th>
<th>$k-\sqrt{kL}$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$C_p$</td>
<td>$C_f$</td>
<td>$C_t$</td>
</tr>
<tr>
<td>Grid 2</td>
<td>20.7</td>
<td>1.8</td>
<td>2.5</td>
</tr>
<tr>
<td>Grid 3</td>
<td>33.8</td>
<td>1.4</td>
<td>1.3</td>
</tr>
<tr>
<td>Grid 4</td>
<td>33.8</td>
<td>1.6</td>
<td>0.8</td>
</tr>
<tr>
<td>Grid 5</td>
<td>30.4</td>
<td>1.5</td>
<td>0.6</td>
</tr>
<tr>
<td>Pereira et al.</td>
<td>0.95</td>
<td>1.7</td>
<td>1.5</td>
</tr>
</tbody>
</table>

The results show that, for the $k-\omega$ based models, a very large uncertainty is predicted for the pressure part of the force while the values obtained for the $k-\sqrt{kL}$ are much lower. It should be also noted that the pressure contribution is relatively small compared to the friction contribution, which shows values between 1 and 3 % for the $k-\omega$ based models and between 3 and 5 % for the $k-\sqrt{kL}$. The uncertainty predicted on the total drag shows that overall, $k-\omega$ SST leads to less uncertainty than $k-\omega$ TNT and $k-\sqrt{kL}$ is placed between these two models.

Compared to the results obtained by Pereira et al., the uncertainties obtained for the $k-\omega$ based models for the pressure are much larger while they are in the same range for the friction. For $k-\omega$ SST, the uncertainty on the total force is also in the same range but they are 4 to 5 times higher for $k-\omega$ TNT. For the $k-\sqrt{kL}$ model, the results are higher for all the components.

Putting these uncertainties in the perspective of the total drag calculated, we see that $k-\omega$ SST leads to the lowest numerical uncertainty, around 2.5 %, but the drag calculated is underestimated. Using $k-\omega$ TNT leads to a total drag prediction closer to the experimental data but the numerical uncertainty, around 9 % on average, is much higher than with the $k-\omega$ SST. The uncertainty obtained with $k-\sqrt{kL}$ is between between the two $k-\omega$ based models with around 4 %, but the drag calculated is the most underestimated.
Figure 6: Experimental results obtained in the towing tank (left) and wind tunnel (right)

Figure 7: Wake flow obtained with $k-\omega$ SST with grids 3 (left) and 5 (right) from the second set

Figure 8: Results obtained by Pereira et al. with $k-\omega$ SST compared to 5th grid of second set with $k-\omega$ SST
4.3 Wake flow

While the focus so far has been the verification of the integral values obtained with the simulations, the wake flow is also a point of interest. When coupling a propeller analysis to the resistance simulations in order to predict the power requirements of the ship, the proper prediction of the wake flow is crucial.

Figure 6 shows the axial velocity contours and transverse velocity vectors obtained during the experiment carried out by Kim et al. [8] and Lee et al. [9] in a towing tank and wind tunnel respectively. The noticeable features of this wake flow are its hook shape and the vortex in the hook.

Figure 7 shows the obtained results with $k-\omega$ SST with grids three and five of the second set. It shows that the bilge vortex is present in the wake but the hook shape is entirely smoothed, in grid three, if not missing, in grid five. Figure 8 shows a comparison of the results obtained by Pereira et al. with the results of this study. In the results of Pereira at al. the hook shape is more visible but still not as pronounced as in the experimental results. Figure 8 also shows that the boundary layer near the symmetry plane is thicker in the present study. The velocities further away of the hook shape are similar.

5 CONCLUSIONS

In this study, a method to obtain trimmed grids as geometrically similar as possible was presented and applied to the flow around the KVLCC2 at model scale Reynold number. Two sets of five grids with different contractions towards the ship were built, and computations using three different turbulence models were carried out. The obtained integral results were analysed using the method proposed by L. Eça and M. Hoekstra to estimate the discretisation error made. The analysis shows that the numerical uncertainty decreases when using grids with a higher contraction towards the ship.

A grid with reasonable density like Grid3, when using the $k-\omega$ SST turbulence model, results in 3% uncertainty on the total drag for the first set and 1.3% for the second set. With $k-\omega$ TNT these values increase respectively to 12% and 8.7%. The uncertainty obtained with the $k-\sqrt{kL}$ model are between the two $k-\omega$ based model with 5.2% with the first set and 3% with the second set.

When taking into account only the difference with the force obtained during the experiments, the $k-\omega$ TNT model performs overall better than $k-\omega$ SST which performs better than the $k-\sqrt{kL}$ model. The $k-\omega$ TNT is on average within 1% of difference with the experimental value, $k-\omega$ SST within 2% to 3% and $k-\sqrt{kL}$ within 4% to 5%. Compared to the results obtained by Pereira et al. for the same exercise on structured grids, the uncertainty obtained in the present study are larger but the difference with the experiment are in the same order of magnitude and show the same trend.

The analysis of the velocity field in the wake of the ship shows that the grids used in this study are able to capture the bilge vortex, but the hook shape visible in the axial velocity field is smoothed out. More details of this hook shape were captured in the results of Pereira et al. with structured grids in combination with the turbulence models used in that study, but the deviation from the experiments are still pronounced.

The results of this study show that the numerical uncertainty and calculated drag are highly
dependent on the turbulence model used. Using unstructured grids results in more uncertainty than using structured grids even though, for the integrated values, the difference with the experiments are still contained in the same order of magnitude. Considering the wake flow prediction, the grids used in this study do not permit to capture the flow details at the same level as the structured grids, but turbulence modelling also plays a major role and needs to be addressed in more detail.

6 ACKNOWLEDGEMENTS

This research is partly funded by the Dutch Ministry of Economic Affairs.

REFERENCES


A GENERALISED UNSTEADY HYBRID DES/BEM
METHODOLOGY APPLIED TO PROPELLER-RUDDER
FLOW SIMULATION

D. CALCAGNI∗, F. SALVATORE∗, G. DUBBIOSO∗, R. MUSCARI∗

∗ Marine Technology Research Institute
National Research Council (CNR-INSEAN)
Via di Vallerano, 139, 00128 Rome, Italy
1e-mail: danilo.calcagni@cnr.it, web page: http://www.insean.cnr.it/

Key words: Propulsion, CFD, Hybrid DES/BEM, propeller-rudder interaction

Abstract. A generalised hybrid viscous/inviscid flow model for the hydrodynamic analysis
of marine propellers is presented. A Boundary Element Method (BEM) to predict propeller
perturbation under inviscid-flow assumptions is combined with a Navier-Stokes solver to describe
the viscous, turbulent flow with propeller effects recast as volume-force terms from BEM. In
the present study, the viscous flow solution is based on a Detached Eddy Simulation (DES)
model valid for unsteady flows. A numerical application is presented by considering a notional
propeller-rudder assembly, and results from the hybrid DES/BEM solution are validated by
comparisons with full DES. The validation study demonstrates the capability of the proposed
hybrid viscous/inviscid flow model to describe transient propeller-induced flow perturbation and
of propeller/rudder interaction in spite of the fact that the geometry of propeller blades is not
resolved but described via a simple and fast volume force model.

1 INTRODUCTION

Computational Fluid Dynamics (CFD) tools based on the numerical solution of the Navier-
Stokes equations have proven the capability to correctly simulate the complex flow features that
characterize the interaction of a rotating propeller with the ship hull and its appendages.

The detailed analysis of the propeller-rudder assembly deserves a specific attention in many
applications. Ship manoeuvring simulations require a careful prediction of rudder loads averaged
over the propeller revolution period with transient flow effects limited to describe the non-
uniform inflow to the propeller. Transient loads induced on the rudder surface by propeller
swirled flow and trailing vorticity become important to identify the occurrence of excessive
pressure pulses on the rudder surface with related risks for erosive cavitation, vibrations, noise
radiation. The global cost of a time-accurate analysis by CFD with resolved rotating propellers
is very high both in terms of mesh generation and computing time of time-marching solutions
with very small time scales associated to propeller rotational speed.
Classical actuator disk approaches, e.g. [18], that are used to limit the computational burden of viscous flow simulations around a fully appended hull can only describe averaged propeller effects and hence cannot be used in problems where transient flow perturbations are of interest. An evolution of actuator disk models is represented by the so-called hybrid viscous/inviscid models in which lifting-line, vortex-lattice [5] [20], Boundary Element Methods (BEM, [4] [12] [16]) solvers to predict propeller induction are combined with Reynolds-Averaged Navier-Stokes Equation (RANSE) solvers to describe the viscous turbulent flow around ship hull and appendages. In spite of more generality with respect to actuator disk models, in most cases the coupling between RANSE and propeller model is limited to steady flow conditions.

Few attempts have been made to fully generalise the approach in order to describe the transient flow perturbation induced by the propeller. Examples are [19], in which the unsteady flow around a podded propeller with ventilation effects is investigated by a hybrid RANSE/BEM model, and [15] in which the transient flow around an isolated propeller in uniform flow is studied by hybrid RANSE/BEM and results are validated by comparing with fully unsteady RANSE.

Aim of the present paper is to follow-up the work in [1], [15] and use the same unsteady viscous/inviscid coupling strategy with RANSE solver replaced by a Detached Eddy Simulation (DES) solver to better describe transient flow features in the propeller wakefield. To the authors’ knowledge this is the first case of a hybrid viscous/inviscid model for ship hydrodynamics in which a DES solver is used. The viscous flow solver is the $\chi$-Navis code, the propeller flow model is the BEM solver PRO-INS, both developed at CNR-INSEAN, see e.g. [3] and [14]. The coupling of the PRO-INS solver with a different RANSE solver is described in [2].

A numerical application of the proposed hybrid DES/BEM model is presented by considering a notional propeller-rudder assembly given by an infinite-span straight rudder placed in the race of the INSEAN E779A model propeller [13]. Numerical results include transient loads on both propeller and rudder, velocity distributions and pressure distributions on the rudder. Predictions by the hybrid model are compared with results by full DES [11] to analyse the capability of the proposed DES/BEM coupling to reproduce transient flow features characterizing the interaction between propeller and rudder.

2 THEORETICAL AND COMPUTATIONAL MODEL

The proposed hybrid viscous/inviscid model extends a formulation presented in [15] in which the velocity field is assumed to be the combination of two contributions: an arbitrary onset flow with velocity $\mathbf{w}$ and the velocity perturbation $\mathbf{v}_P$, induced by a propeller immersed in it. Propeller flow is studied by a boundary element model for inviscid flows, while a Navier-Stokes solver is used to describe the onset flow under general viscous flow conditions.

A Reynolds-Averaged Navier-Stokes model is considered in [15], whereas Detached Eddy Simulation (DES) is addressed here. The boundary integral formulation for inviscid flows is valid for an isolated propeller in unbounded flow. Propeller-rudder interaction is described through the coupling between propeller flow and onset flow problems, as explained later.

2.1 Inviscid flow model by BEM

Assuming an inviscid, irrotational propeller-induced flow, perturbation velocity can be described in terms of a scalar velocity potential, $\mathbf{v}_P = \nabla \varphi$. Under incompressible flow assumptions,
mass and momentum equations yield that the velocity potential \( \phi \) satisfies the Laplace equation and the pressure \( p \) follows from the Bernoulli equation, respectively

\[
\nabla^2 \phi = 0, \quad \frac{\partial \phi}{\partial t} + \frac{1}{2} \| \nabla \phi + \mathbf{v}_I \| + \frac{p}{\rho} + g z_0 = \frac{1}{2} \| \mathbf{v}_I \|^2 + \frac{p_0}{\rho},
\]

(1)

where \( \rho \) is water density, \( p_0 \) is the free-stream reference pressure and \( g z_0 \) is the hydrostatic head. Quantity \( \mathbf{v}_I = \mathbf{w} + \Omega \times \mathbf{x} \) denotes the inflow to the propeller as observed from a frame of reference \((Ox_Fy_Fz_F)\) (Fig. 1(a)) fixed with the propeller rotating at angular velocity \( \Omega \), while \( \mathbf{w} \) is the onset flow velocity, to be studied under viscous-flow assumptions, as described later.

Boundary conditions for \( \phi \) are obtained by imposing vanishing perturbation at infinity and impermeability on the propeller surface, that is \((\mathbf{v}_p + \mathbf{v}_I) \cdot \mathbf{n} = 0\), or

\[
\frac{\partial \phi}{\partial n} = - (\mathbf{w} + \Omega \times \mathbf{x}) \cdot \mathbf{n}
\]

(2)

where \( \mathbf{n} \) is the unit normal to the surface. Potential flow theory applied to lifting bodies implies the introduction of a potential-discontinuity surface, the trailing wake \( S_W \), where vorticity generated on blades is shed downstream. An analytical helicoidal wake surface model is used here.

Following [7], the Laplace equation for \( \phi \) is solved by a boundary integral representation as

\[
E(x) \phi(x,t) = \int_{S_B} \left( \frac{\partial \phi}{\partial n} G - \phi \frac{\partial G}{\partial n} \right) dS - \int_{S_W} \Delta \phi_{TE} (t - \tau) \frac{\partial G}{\partial n} dS
\]

(3)

where \( S_B \) is the propeller surface, \( t \) is time, \( \Delta \phi_{TE} \) denotes potential discontinuity at blade trailing edge where the wake is shed and \( \tau \) is the shedding delay associated to wake points (Kutta condition). Quantities \( G, \partial G/\partial n \) are unit source and dipole in the unbounded three-dimensional space, and \( E(x) \) is a field function to distinguish the case where \( x \) is inside the flow field \((E = 1)\) or on the solid boundary surface \((E = 1/2)\).

Recalling \( \mathbf{v}_p = \nabla \phi \) and Eq. (2), propeller induced velocity may be evaluated from Eq. (3) through a linear problem in which unknowns are located on the propeller surface.

The numerical solution of Eq. (3) is obtained here by a low-order Boundary Element Method implemented into the PRO-INS code developed by CNR-INSEAN. Once \( \phi \) on \( S_B \) is known, the pressure \( p \) is determined from the Bernoulli theorem, and hydrodynamic forces acting on propeller blades follow as

\[
f = \sum_{n=1}^{Z} \int_{S_{Bn}} (-p \mathbf{n} + \tau) dS,
\]

(4)

where \( S_{Bn} (n = 1,...Z) \) denotes the surface of the \( n \)-th blade of a \( Z \)-bladed screw. Using a common approach in inviscid-flow models, the distribution of viscosity–induced tangential stress \( \tau \) is estimated by using expressions that are valid for flat plates in laminar and turbulent flow at same Reynolds number as blade sections.
2.2 Viscous flow model by DES

The viscous flow surrounding the propeller is evaluated by integration of the Navier-Stokes equations for unsteady incompressible flows:

\[
\nabla \cdot \mathbf{v} = 0 \tag{5}
\]

\[
\frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla) \mathbf{v} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{v} + \mathbf{B}
\]

where quantity \( \mathbf{B} \) denotes a generic source term. In the present work, it represents the propeller induced forces predicted by BEM.

The numerical algorithm to solve Eq. (5) is implemented into an in–house solver based on a finite volume technique with pressure and velocity co-located at cell center. Viscous terms are integrated by a standard second order centered scheme, whereas for the convective and pressure terms a third order upwind scheme is chosen. Of course, because of the treatment of the viscous terms, the scheme remains formally second order in space. The turbulence closure is based on the Detached Eddy Simulation (DES) version of the Spalart-Allmaras turbulence model [17].

The physical time derivative in the governing equations is approximated by a second-order accurate, three-point backward finite difference formula [3]. In order to obtain a divergence free velocity field at every physical-time step, a pseudo-time derivative is introduced in the discrete system of equations [6]. As it will be explained in the following, every \( n \) pseudo-time steps of the inner iteration (\( n \) being a parameter chosen through numerical tests) the BEM algorithm is invoked in order to update the \( \mathbf{B} \) term in Eq. (5).

No-slip boundary conditions are enforced at solid walls. At the inlet boundary the velocity is set to the undisturbed flow value, whereas at the outflow the pressure is set equal to zero.

The mesh built to discretize the fluid domain is composed by structured, partially overlapping blocks, and a chimera algorithm is used in order to interpolate the solution among different chimera sub-grids, [8] [9] [10].

The introduction of propeller effects via viscous/inviscid coupling is obtained in a body force fashion. Specifically, propeller blades are not represented as solid boundaries of the computational domain and a cylindrical grid block is placed to fill the propeller region. Volume forces describing propeller induction and predicted by BEM, as shown later, are distributed over this grid block and added as source term \( \mathbf{B} \) to the right-hand side of the momentum equations.

2.3 DES/BEM coupling

The viscous/inviscid coupling strategy in [15] is used here and briefly recalled for the sake of completeness. Viscous and inviscid-flow solutions are integrated through an interface in which two quantities are exchanged:

- volume-forces \( \mathbf{B} \) recasting hydrodynamic forces on propeller blade surface by BEM as a volume force distribution in the Navier-Stokes equations;

- effective-inflow \( \mathbf{w} = \mathbf{v} - \mathbf{v}_p \): rotational onset flow velocity to impose in BEM the impermeability condition on blades by using Eq. (2).
Volume forces $B$ are derived by a time-accurate procedure that preserves spatial distribution and position in time of hydrodynamic forces exchanged between fluid and propeller blades.

At each time step, loading $(-p\mathbf{n} + \tau)$ (Eq. (4)) is pointwise averaged on blade suction and pressure sides and associated to the actual position of the blade mean surface $\hat{S}_{B_k}$. The resulting force distribution is referred to a cylindrical coordinate system $(Ox\tau\theta)$, Fig. 1(a) and can be written as $b = b(x, r, \theta, t)$. By definition, $b$ is zero at points $(x, r, \theta)$ outside $\hat{S}_{B_k}$. Recalling that $b$ represents a time-dependent Dirac distribution along coordinate $x$, an equivalent volume distribution can be easily obtained by introducing a function $\xi = \xi(x)$

$$\int b(x, r, \theta, t)dx = b_0(r, \theta, t) \int \xi(x)dx \quad (6)$$

In the present work, $\xi = \xi(x)$ is a normalised Gaussian distribution. For each value of $r, \theta$, the function is centered at point $x$ identifying the position of surface $\hat{S}_{B_k}$ at time $t$. An example of the Gaussian distribution is shown in bottom right Fig. 2 below. The steepness of the Gaussian function controls the thickness in $x$ direction of the region where volume forces replacing blade effects are imposed. In the present study, the distribution is chosen to have that the integral in the right-hand side of Eq. (6) is 0.99 when the integration interval along $x$ equals 0.075 propeller diameters.

The coupling between viscous and potential-flow solutions is achieved within the inner iteration of the dual-stepping procedure anticipated in section 2.2.

At the initial pseudo-time step, the viscous onset flow is obtained by the solution of Eqs. (5) with zero volume forces $B$ describing propeller effects. The resulting velocity distribution $v$ (which is not even a soleinodal field, as convergence in the inner iteration is not reached) is used to evaluate the inflow to the propeller as $w = v$. A first estimate of the propeller-induced perturbation is obtained by using $w$ to impose boundary conditions by Eq. (2) and solve BEM equations, and a first guess of volume force distribution $B$ follows.
From this point, the inner integration in the pseudo-time is started and convergence to a soleinodal field is reached together with convergence of the viscous–potential flow solutions. We found that it is uselessly burdensome update $B$ at each pseudo-time step, and that the best compromise was to call the BEM solver every $O(10)$ pseudo-time steps. In this way, every new estimation of $B$ predicted by BEM is not too different from the previous one and the convergence of the inner iteration of the viscous solver is not hindered.

The unsteady-flow BEM solution is synchronised with propeller rotation in the viscous flow solution, except at the first time step, when the unsteady BEM solution is initialised by marching in time for a number of revolutions to achieve a correct description of transient flow contributions from the trailing wake, see Eq. (3).

As already mentioned, the BEM used here describes an isolated propeller, and hence no rudder surface contribution is explicitly given in Eq. (3). Presence of the rudder in the potential flow solution is accounted for through the effective inflow $w$ where rudder perturbation to the flow field is described by the viscous flow solution of Eq. (5) with propeller perturbation included via source terms. An approximation in the evaluation of the effective inflow $w$ is due to the fact that velocity contributions $v$ and $\nabla \phi$ are evaluated not on the actual propeller surface but over a plane perpendicular to the propeller axis corresponding to the upstream boundary of the volume force grid block.

3 NUMERICAL APPLICATION

A numerical application of the hybrid DES/BEM methodology is presented here by considering the interaction between a propeller and a rudder in its race under uniform onset flow conditions. Results by hybrid DES/BEM are validated against full-DES calculations using the methodology described in [11]. This study extends the analysis in [15], where an isolated propeller in uniform flow is addressed and hybrid unsteady RANSE/BEM results are compared with full-RANSE results. In both cases, the four-bladed INSEAN E779A model propeller, with diameter $D = 227.27$ mm is considered (see [13] for details).

A notional propeller-rudder assembly is analysed here in which the rudder is given by a straight cylinder with NACA 0020 sections aligned to the propeller axis and span extending to the farfield boundary of the computational domain. Rudder chord is 1.58$R$ and leading edge is placed at distance 1.18$R$ downstream the propeller plane. A sketch of the propeller-rudder assembly is given in Fig. 1(b).

Table 1 summarizes main geometry parameters and operating conditions considered in the numerical study. The Reynolds number is defined as $Re = \pi n D^2 / 2\nu$, by considering propeller radius $R = D/2$ as the reference length and velocity of blade tip as the reference speed, $V_{ref} = 2\pi n R$. The non-dimensional revolution period is then $T = 2\pi$. Considering inflow speed $V_0 = 5$ m/s and rotational speed $n = 25$ rps, propeller operates at advance coefficient $J = V_0/nD = 0.88$, that corresponds to the design point for the INSEAN E779A screw.

The computational domain for the numerical solution of the DES equations is delimited by a cylinder co-axial to propeller axis, with diameter of 50 $D$, axial length of 50 $D$ and inlet/outlet planes at 25 $D$ from the propeller plane. Full-DES computations are performed on three grid levels. The finest grid consists of 18.9M cells, with 2.4M discretizing the fluid domain surrounding blades, and coarser grids are obtained by removing every other point from the finer level. Using
Table 1: Propeller-rudder case study: geometry and operating conditions

<table>
<thead>
<tr>
<th>Parameter</th>
<th>INSEAN E779A propeller</th>
<th>Rudder</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blades</td>
<td>4</td>
<td>Section</td>
</tr>
<tr>
<td>Diameter</td>
<td>0.227 m</td>
<td>Chord</td>
</tr>
<tr>
<td>Pitch ratio (0.7R)</td>
<td>1.183</td>
<td>Span</td>
</tr>
<tr>
<td>Expanded area ratio</td>
<td>0.468</td>
<td>Deflection angle</td>
</tr>
<tr>
<td>Hub ratio</td>
<td>0.295</td>
<td>Distance LE-Prop. plane</td>
</tr>
<tr>
<td>Diameter</td>
<td>0.227 m</td>
<td>Chord</td>
</tr>
<tr>
<td>Pitch ratio (0.7R)</td>
<td>1.183</td>
<td>Span</td>
</tr>
<tr>
<td>Expanded area ratio</td>
<td>0.468</td>
<td>Deflection angle</td>
</tr>
<tr>
<td>Hub ratio</td>
<td>0.295</td>
<td>Distance LE-Prop. plane</td>
</tr>
</tbody>
</table>

Operating conditions

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Onset flow velocity, $V_0$</td>
<td>5 m/s</td>
</tr>
<tr>
<td>Propeller rate of revolution, $n$</td>
<td>25 rps</td>
</tr>
<tr>
<td>Reynolds number, $Re$</td>
<td>$1.78 \times 10^6$</td>
</tr>
<tr>
<td>Advance coefficient, $J$</td>
<td>0.88</td>
</tr>
</tbody>
</table>

In the hybrid DES/BEM model, the volume grid is the same of the full-DES case except for blade-fitted blocks replaced by the volume-force block. This yields a saving of 2.2M cells on the finest grid with respect to full-DES. Details of grid blocks surrounding the propeller and the rudder are shown in Fig. 2, where differences between grids used for full-DES (left) and for hybrid DES/BEM (right) are apparent. In particular, blade-fitted blocks necessary for capturing blade boundary layer in the full-DES analysis are replaced by a simple torus filling the volume spanned by rotating blades. For the inviscid-flow BEM solution, each blade surface is discretized into 864 elements and the hub surface into 624 elements. The trailing wake surface (see Eq. 3) extends for 3 revolutions with 3240 elements per revolution.

Both hybrid and full DES simulations are time-marching to describe the swirled flow induced by the rotating propeller to the rudder downstream. In full-DES calculations, propeller and hub blocks rotate with respect to fixed blocks built around rudder and background. Time stepping corresponds to 1 deg propeller rotation. In the hybrid DES/BEM simulation, the whole volume grid is fixed to the rudder and the hub surface is non-rotating. Only the surface grid used to discretize blades and hub in the BEM solution rotates, with time discretization of 2 deg per step.

Bottom right picture in Fig. 2 shows the volume force block and the distribution of volume force terms over a longitudinal axial plane. It may be noted that the inner boundary of the torus is not fitted to the tapered hub surface. As a consequence, there is a thin layer at blade root where no volume terms are defined to simulate the presence of the blade. The impact of differences between grids and boundary conditions at the hub is discussed in the following pages.

In order to compare results of numerical solutions by hybrid DES/BEM and by full DES, it is fundamental to verify that the hydrodynamic loading generated on propeller blades is consistent between the two solutions. Specifically, thrust $T$ and torque $Q$ obtained by integrating normal and tangent stress by BEM are to be compared with corresponding quantities from the full DES solution. In addition to this, thrust and torque obtained by integrating volume force terms are also to be verified to quantify possible errors related to recasting surface loads by BEM as volume loads. Transient loads using these different definitions are plotted in Fig. 3, while averaged values over a propeller revolution period are collected in Table 2. For the sake of completeness, thrust and torque contributions evaluated on the hub surface and axial force on the rudder $F_x$ are also given. Recalling the rudder has a theoretically infinite span, in the present analysis loads are obtained by integrating over a surface with extension of 1.3 $D$ and centered at propeller axis.
Figure 2: Discretization of the fluid region around propeller and rudder over a horizontal plane at \( z = 0 \): full DES (left) and hybrid DES/BEM (right). Top: propeller and rudder. Bottom: details of grid blocks around the hub surface.

The following definitions of thrust, torque and axial force coefficients are used

\[
K_T = \frac{T}{\frac{1}{2} \rho n^2 D^4}, \quad K_Q = \frac{Q}{\frac{1}{2} \rho n^2 D^5}, \quad K_{Fx} = \frac{F_x}{\frac{1}{2} \rho n^2 D^4}.
\]  

(7)

Left Fig. 3 shows that thrust and torque on blades obtained as volume force integral in the hybrid DES/BEM model overestimate full-DES predictions, with differences of time-averaged values of 6.8\% for \( K_T \) and 4.9\% for \( K_Q \), see Table 2. Force fluctuations \( \Delta K_T \) and \( \Delta K_Q \) by hybrid DES/BEM are sensibly higher than those predicted by full DES. Volume forces are lower than surface loads by BEM, with negligible differences for thrust while torque reduces of 4.2\%.

Axial forces on the hub surface and on the rudder portion are plotted in right Fig. 3. Differences between hybrid DES/BEM and full DES predictions of hub thrust are small but since averaged values are very small, a high percentual error of 70\% is found. This can be explained recalling that, as discussed above, the hub region is where full DES and hybrid DES/BEM computational set-ups present main differences.
Figure 3: Time histories of propeller-rudder system loads, J=0.88. Left: thrust and torque on propeller blades. Right: axial force on rudder and hub.

In contrast to this, it is interesting to observe that time histories of axial force on the rudder evaluated by the two models are in fair agreement, with hybrid model result only 6.8% lower than DES. This can be considered as a global indicator that full DES and hybrid DES/BEM provide comparable descriptions of propeller induction to the flow field surrounding the rudder. A deeper analysis of this aspect is the objective of the present study and to this aim, velocity and pressure fields downstream the propeller are investigated.

Table 2: Propeller-rudder system loads, J=0.88. Time-averaged loads and fluctuations on propeller blades, hub and rudder.

<table>
<thead>
<tr>
<th>Non-dim. loads</th>
<th>Full DES</th>
<th>Hybrid DES/BEM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Propeller (blades only)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$K_T$ (average)</td>
<td>0.146</td>
<td>0.156</td>
</tr>
<tr>
<td>$10K_Q$ (average)</td>
<td>0.305</td>
<td>0.320</td>
</tr>
<tr>
<td>$\Delta K_T/K_T$</td>
<td>2.1%</td>
<td>7.7%</td>
</tr>
<tr>
<td>$\Delta K_Q/K_Q$</td>
<td>1.3%</td>
<td>6.9%</td>
</tr>
<tr>
<td>Rudder</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$K_{Fx}$ (average)</td>
<td>$-6.59 \times 10^{-4}$</td>
<td>$-6.14 \times 10^{-4}$</td>
</tr>
<tr>
<td>Hub</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$K_T$ (average)</td>
<td>$-1.22 \times 10^{-4}$</td>
<td>$-0.35 \times 10^{-4}$</td>
</tr>
<tr>
<td>$10K_Q$ (average)</td>
<td>$-2.33 \times 10^{-4}$</td>
<td>$-0.53 \times 10^{-4}$</td>
</tr>
</tbody>
</table>

Axial velocity distributions on transversal planes downstream the propeller are shown in Fig. 4 at time step corresponding to blade in the reference position $\theta = 0$. Planes are located downstream the region swept by blade trailing edges ($x/R = 0.18$), upstream the rudder leading edge ($x/R = 1.125$), and downstream the rudder trailing edge ($x/R = 3.5$), Fig. 1(b). The black circle in bottom left Fig. 4 indicates the outer boundary of the volume force block.
A qualitative comparison highlights a good agreement between results from the two approaches, with the solution based on propeller blades replaced by volume force terms inducing smoother distributions of axial velocity than the fully resolved solution. While such a trend is expected, it is interesting to observe that at plane \( x/R = 1.125 \) just downstream the volume force grid block, the hybrid model determines a velocity distribution that is consistent with the full DES solution. As expected for reasons discussed above, larger differences are observed in the hub region. Moving downstream, the agreement between hybrid DES/BEM and full DES solutions increases. In particular, the two solutions reveal the same mechanisms of interaction between propeller-induced flow and rudder surface. Vortical structures shed from blade tips and hub are splitted at the rudder leading edge in counter-rotating vortices. The resulting velocity field near the rudder surface yields to a composition of structures with the streamtube deformed and vertically shifted in opposite directions on starboard and portside. This is apparent on the plane downstream the rudder trailing edge (right Fig. 4), with distance between vortices tending to increase on the starboard side below the shaft and on the portside above the shaft. High velocity gradients are induced on the vertical midplane \( y = 0 \).

Quantitative comparisons of predicted velocity distributions are presented in Figs. 5 and 6, for three selected radial locations over the two planes downstream the propeller \((x/R = 0.33)\) and rudder trailing edge \((x/R = 3.5)\), Fig. 1(b). Distributions of the axial velocity and of the intensity of in-plane velocity are shown at distances from propeller axis \( r/R = 0.24, 0.75, 1.0 \). As expected, the largest discrepancy between solutions is found at sections near the hub.
The hub swirl effect is not taken into account in the hybrid approach and the propeller root representation by body forces is influenced by the thin gap between the tapered hub and the untapered volume force block, see Fig. 2. The agreement becomes good at outer radii just downstream the propeller (Fig. 5) and downstream the rudder (Fig. 6). Comparable velocity distributions are obtained both at $r/R = 0.75$ and at $r/R = 1.0$ where the effect of blade-shed
tip vortices dominates. As already noticed, velocity fluctuations are lower in hybrid DES/BEM results than in the full DES ones.

Further insight of flow field predictions are obtained by considering velocity distributions at vertical plane $y/R = 0.05$ and at horizontal plane $z = 0$. Contour maps of the axial velocity component are shown in Fig. 7. The black lines in the right pictures indicate upstream and downstream boundaries of the volume force block used by the hybrid model.

A limited agreement between results in the hub region is clearly shown, whereas the two solutions match very well in the other parts of the fluid domain and reveal the same flow features related to the propeller-shed wake and its interaction with the rudder. Different pressure distributions on rudder surface at starboard and portside influence the velocity distribution, with vortical structures impinging the rudder that are splitted and shed with different velocity on the two sides. This results into an asymmetric velocity distribution, here visible on vertical planes, top Fig. 7, that depends on the sense of rotation of the propeller. In the present case, with a right-handed screw, stronger axial velocity perturbation is found below the propeller axis line at starboard, while an opposite condition occurs at portside (not shown here), with stronger axial velocity perturbation above the propeller axis line.

**Figure 7**: Contour map of the axial velocity field on longitudinal planes, propeller angular position $\theta = 0^\circ$. Left: full DES. Right: hybrid DES/BEM. Top: vertical plane $y/R = 0.05$. Bottom: horizontal plane $z/R = 0$. 
Pressure distributions on rudder portside for three time steps corresponding to blade angular positions $\theta = 0, 30, 60$ degrees are shown in Fig. 8. The agreement between hybrid DES/BEM and full DES solutions is very good, with fully comparable positive and negative pressure peaks in the leading edge region. This is confirmed by quantitative comparisons in Fig. 9 where chordwise pressure distributions on sections at $z/R = 0.75$ and 1.0 above the shaft line are plotted for a representative time step ($\theta = 0$). Small differences in predicted pressure distributions occur only in a narrow band where the hub vortex impinges the rudder leading edge. This may be detected in Fig. 8, and is quantified in Fig. 10 for a chordwise section at $z/R = 0.24$ above the shaft line and propeller positions $\theta = 0^\circ$ and $90^\circ$. Left and right plots in Fig. 10 refer to positions separated by 90 degrees that for the present four-bladed propeller corresponds to the blade passing period. It is interesting to observe that while the hybrid DES/BEM solution is nearly periodic with blade passing frequency, fluctuations are apparent in the full DES solution. Such a different behaviour can be explained with a general trend of the hybrid DES/BEM model to smooth out propeller-induced perturbations that can be captured when blades are actually described as solid boundaries.

Main findings from the present comparative analysis between hybrid DES/BEM and full DES solutions can be summarised as follows:

- velocity fields predicted by the hybrid DES/BEM model are in agreement with those obtained from the full DES solution. This includes flow features like the trailing vorticity pattern in the propeller wake and its interaction with the rudder;

- a non-trivial result is that just downstream the fluid region where volume force terms are imposed to describe propeller induction, the solution by the hybrid DES/BEM model
agrees with the fully resolved DES solution;

- the hybrid DES/BEM model fails to capture high-frequency perturbations that are captured when blades are actually described as solid boundaries in the full DES solution.

Figure 9: Chordwise pressure distribution on rudder sections $z/R = 0.75$ (left) and $z/R = 1.0$ (right). Propeller angular position $\theta = 0^\circ$.

Figure 10: Chordwise pressure distribution on rudder section $z/R = 0.24$ at propeller angular positions $\theta = 0^\circ$ (left) and $\theta = 90^\circ$ (right).

4 CONCLUDING REMARKS

A generalised hybrid viscous/inviscid flow computational model for the simulation of propeller-induced flows has been presented and applied to the analysis of a propeller-rudder assembly. The methodology combines a Boundary Element Method (BEM) to predict propeller perturbation under inviscid-flow assumptions and a Detached Eddy Simulation (DES) Navier-Stokes solver to describe the viscous, turbulent flow around the rudder with propeller effects recast as volume-force terms from the BEM solution. The coupling between BEM and DES solvers is achieved via a time-marching iterative procedure to account for transient flow perturbation induced by a rotating propeller. The resulting computational model generalizes the concept of actuator disk used in ship hydrodynamics where only time averaged global propeller effects are described. The
model also generalizes existing hybrid RANSE/BEM models by coupling BEM with an unsteady DES Navier-Stokes solver to better describe transient turbulence and vorticity dynamics in the propeller wakefield.

Numerical results present hydrodynamic forces generated on propeller and rudder, pressure distributions on the rudder, velocity and vorticity downstream the propeller, and predicted by the hybrid DES/BEM model and validated by comparison with full DES solutions. Results of this validation study demonstrate the capability of the proposed hybrid viscous/inviscid flow model to reproduce full DES descriptions of propeller-induced perturbation to the rudder and the surrounding flow in spite of the fact that propeller blades are not resolved but simply described via volume forces by BEM. In particular, the analysis reveals a quantitative agreement between results from hybrid BEM/DES and full DES. Even at very small distance downstream the region where blades are replaced by volume forces, the hybrid model captures flow features with DES accuracy.

These findings may have a clear impact on applications of the proposed methodology to ship hydrodynamics problems where the transient flow perturbation induced by rotating propulsors is the objective of the study, as e.g., to investigate propeller-rudder interaction, pressure pulse radiation to the hull plate, hydroacoustic noise emission, propeller action during manoeuvers. Numerical results from the present study show areas of further work necessary to improve the consistency of propeller force predictions by the BEM model and to generalise the shape of the volume force block to fit tapered propeller hubs.

REFERENCES


EXTENSION OF THE TURBULENT SPOT METHOD TOWARDS ARBITRARY REYNOLDS STRESSES AND INTEGRAL LENGTHS

HANNES KRÖGER*, NIKOLAI KORNEV† AND PASCAL ANSCHAU**

*Chair of Modeling and Simulation (LEMOS)
University of Rostock, Albert-Einstein-Str. 2, 18059 Rostock, Germany
e-mail: hannes.kroeger@uni-rostock.de, web page: http://www.lemos.uni-rostock.de/

†Chair of Modeling and Simulation (LEMOS)
e-mail: nikolai.kornev@uni-rostock.de

**Schiffbau-Versuchsanstalt Potsdam GmbH
Marquardter Chaussee 100, 14469 Potsdam, Germany
e-mail: anschau@sva-potsdam.de, web page: http://www.sva-potsdam.de/

Key words: Large-Eddy simulation, inflow boundary condition, turbulence synthesis

Abstract. The paper presents an extension of the Turbulent Spot method which enables to obey the continuity of the fluctuations while producing arbitrarily high anisotropy at the same time. The derivation of the structures is summarized and expressions for their Reynolds stresses and length scales are presented. Finally, the newly derived structures are applied to a turbulent channel flow simulation and compared with other means of turbulence synthesis.

1 INTRODUCTION

In many engineering problems, accuracy of the Reynolds averaged Navier Stokes (RANS) simulations is not sufficient and a more detailed information on the properties of turbulence are required than these which RANS solution provides. A possible remedy is to employ scale resolving techniques like Direct Numerical Simulation (DNS) or Large Eddy Simulation (LES). While DNS still remains out of reach, LES has become a viable option even for industrial users due to constantly increasing power of modern computers. In this context, an important development during the last decades are hybrid RANS/LES methods which extend the applicability of scale resolving methods far towards high Reynolds number flows by excluding the expensive near-wall region from LES and applying unsteady RANS there.

The domain in a flow simulation is always bounded by in- and outlets. Proper boundary conditions at these boundaries have to be prescribed in a CFD simulation of the flow device,
especially at the inlet. In common engineering practice, the flow through the inlet is already turbulent. In RANS context, mean quantities and integral properties of the turbulent fluctuations then have to be prescribed. In contrast, scale resolving simulation techniques gain their advantage by directly including the turbulent fluctuation velocities in the solved velocity field. Thus, the turbulent content at inlet boundaries has to be explicitly prescribed in terms of an unsteady velocity field.

Formulation of the inlet condition is a well-recognized topic in LES and DNS research. Overview of existing methods can be found e.g. in [18, 15, 6, 5, 16, 12]. According to Schlüter et al. [15] the methods for specification of turbulent inlet conditions utilize one of four following techniques:

1. natural laminar turbulent transition,
2. random uncorrelated oscillations,
3. LES or DNS auxiliary simulation with periodic boundary conditions in a domain in front of the area of interest,
4. synthetic turbulent fields.

First two techniques are nowadays not used because the first one requires huge computational resources whereas the second one generates uncorrelated fields which quickly dissipate behind the inlet.

The third approach is widely used for flows with any dominating flow direction. In many cases, when the flow domain is a continuation of pipe or channel flows, the use of periodic boundary conditions in auxiliary domain is the most efficient way to generate inlet conditions. For flows with change of averaged parameters in auxiliary simulation domain, such as, for instance, boundary layers, Spalart proposed the recycling method [17]. In this case the modified periodic boundary conditions are enforced in which the data from outlet to inlet are copied with some rescaling factor. Extension of this technique to three dimensional flows is quite difficult because the recycling direction is hard to detect. A proper development of perturbations in auxiliary domain requires a certain length. To reduce this length, a special forcing within the auxiliary domain is introduced in [1] and [13]. The forcing term is added as an additional body force to control the integral parameters of the boundary layer evolving in the auxiliary domain. The biggest weakness of the third approach is the complexity of its application for arbitrary geometries.

Within the last approach the turbulence is artificially generated without solution of flow equations. The task is to synthesize a turbulent velocity field \( \mathbf{U}(x, t) \):

\[
\mathbf{U}(x, t) = \overline{\mathbf{U}}(x) + \mathbf{u}(x, t)
\]

where \( \overline{\mathbf{U}}(x) \) is the mean velocity which is supposed to be known. The fluctuations \( \mathbf{u} \) need to have a number of properties, which we, according to our experience, list below in the order of their importance:

1. \( \mathbf{u} \) should be spatially and temporally correlated.
2. They have to possess prescribed Reynolds stresses \( R_{ij} = \overline{u_i u_j}(x) \).
3. Additionally to the requirement 1, \( \mathbf{u} \) has to possess prescribed integral lengths \( L_{ij}(x,e_\eta) = \int_0^\infty \rho_{ij}(x,\eta)e_\eta \, d\eta \).

4. \( \mathbf{u} \) should fulfill the continuity constraint \( \nabla \cdot \mathbf{u} = 0 \).

5. Additionally to the requirements 1 and 3, \( \mathbf{u} \) should have prescribed correlation functions

\[
r_{ij}(x,\eta) = \frac{u_{i}(x,t)u_{j}(x+\eta,t)}{u_{i}(x,t)u_{j}(x,t)}
\]

or prescribed spectra.

Artificial turbulence always needs a certain distance (adaptation distance) to evolve into real turbulence. Violation of these requirements can lead to an extension of adaptation distance.

The requirement 4 is especially important for applications where a volumetric oscillation field is necessary. For the plane inlet, the derivative perpendicular to the inlet can make sense by utilization of the Taylor hypothesis. It is assumed that the inlet is far from the area of strong flow change and the turbulence can be considered as a frozen one close to the inlet. Our experience shows that the violation of req. 4 can lead to a weak convergence of the iterative process for solution of the flow equations or artificial pressure fluctuations (i.e. noise) in the flow domain. Usually, in existing methods only the requirements 1 and 2 are satisfied. To fulfill the req. 5, either the correlation function or spectrum should be specified which are usually not available for complex flows. The condition 3 thus represents a relaxed constraint where not the shape of correlation function needs to be known but only its integral.

The current work deals with a method for turbulence synthesis with the special focus on fulfillment of the continuity constraint by retaining arbitrary anisotropy in the Reynolds stresses. We propose the extension of the turbulent spot method based on exact mathematical solutions which fulfills the requirements 1-4. In the next section we briefly outline the mathematical basis of the method.

2 TURBULENT SPOT METHOD

2.1 Non-solenoidal version of the method

The initial version of the method has been developed and published in 2003 for inhomogeneous non-solenoidal turbulent fields [7]. The starting point for the development of the turbulent spots (TS) method were works in which turbulence was simulated by a set of stochastically distributed vortex structures.

The turbulent field in TS method was represented as a set of \( N \) compact spots with an inner primary velocity distribution with components \( v^n_i(r - r^i) = \varepsilon^n_i f_n(r - r^i) \), where \( n \) is the component number, \( r^i \) is the spot center, \( \varepsilon^n_i \) is a random number, uniformly distributed between \(-1\) and \(1\), and \( i = 1, N \). The velocity fluctuation at any point \( r \) is equal to the sum of contributions of all spots \( \mathbf{v} = \sum_{i=1}^N v^i(r - r^i) \). The stochastic behavior of the velocity field is provided by stochastic distribution of the spots centers in space and random choice of the vector \( \varepsilon^i \). If the velocity field \( \mathbf{v} \) moves through any inlet surface, which is not necessary plain, with the unit velocity \( W_1 = 1 \) in, say, \( x_1 \) direction, the \( x_1 \) coordinate becomes equal to time, \( x_1 = Wt \). This is an application of the Taylor hypothesis often used in the turbulence research. If the signs of components of the vector \( \varepsilon^i \) are chosen statistically independently, then all one-point correlations, calculated at any fixed observation point at the inlet, satisfy the condition.
\( \overline{v_n} v_k = \delta_{nk} \sum_{i=1}^{N} \overline{v_i^n v_i^k} \), where overline stands for the time averaging. If a spot has a compact non zero support, the field \( v \) is spatially (and temporally) correlated since two point correlations \( v_n(r) v_k(r + \eta) = \delta_{nk} \sum_{i=1}^{N} v_i^n(r) v_i^k(r + \eta) \) are not zero. Therefore, it is possible to generate the field with specified spectra (or autocorrelation function) and integral length. In this form the TS method is theoretically identical to the Synthetic Eddy Method which appeared later in [3].

2.2 Consideration of anisotropy in synthetic turbulence generators.

Since the most of turbulent flows are strongly anisotropic, the synthetically generated signal should possess the prescribed Reynolds stresses. An elegant way to consider the anisotropy was proposed in [10]. Once three components \( v_i, i = 1, 3 \) of primary velocity are generated separately and conditioned \( v_{(i)} v_{(i)} = 1 \), the secondary velocity field is calculated from the linear combination

\[
 u_i = a_{ij} v_j \tag{2}
\]

where \( a_{ij} \) is the matrix which can be found from the condition \( R_{ij} = u_i u_j = v_n v_k a_{im} a_{jk} \). If \( v_{(i)} v_{(i)} = 1 \) and \( v_i v_{j \neq i} = 0 \) the matrix satisfying \( R = aa^T \) can be found from the Cholesky algorithm

\[
 a_{ij} = \begin{pmatrix}
 \sqrt{R_{11}} & 0 & 0 \\
 R_{21}/a_{11} & \sqrt{R_{11} - a_{21}^2} & 0 \\
 R_{31}/a_{11} & (R_{32} - a_{21} a_{31})/a_{22} & \sqrt{R_{33} - a_{31}^2 - a_{32}^2}
 \end{pmatrix}
\tag{3}
\]

Two aspects should be noted when applying this transformation: First, the cross-correlations of the primary velocities has to be zero: \( \rho_{ij} = u_i u_j = 0 \). This is e.g. not fulfilled for the DSRFG turbulence generators while it is fulfilled for TSM and SEM. Second, the transformation affects the length scales and the spectrum of the produced fluctuations. The authors derived a modified transformation which preserves the autocorrelation functions [8].

2.3 Incorporation of divergence free condition.

A non-solenoidal character of the generated fluctuations \( \nabla \vec{u} \neq 0 \) was a big drawback to the turbulent spot method. Later on, the method was extended to obey the continuity condition \( \nabla \cdot \vec{u} = 0 \) by deriving the inner velocity distribution from a vector potential \( A \) [9]. The idea is based on the fact that the velocity field obtained as \( \vec{u} = \nabla \times A \) satisfies the continuity condition \( \nabla \cdot \vec{u} = \nabla (\nabla \times \vec{A}) = 0 \) automatically.

Essentially the same idea was proposed later by Poletto et al [14] who used the Biot-Savart kernel which is obtained from the condition \( \vec{u} = \nabla \times \vec{A} \) with the vector potential \( \vec{A} \) taken as the fundamental solution of the Poisson equation \( \Delta \vec{A} = -\nabla \times \vec{u} \). We called this new spots of vector potential with corresponding divergence-free velocity fields as "vortons" following the term introduced by Saffman.

Unfortunately, the vortons, introduced above, are ideal for isotropic turbulence and difficult to apply for anisotropic flows close to the wall where the Reynolds stress \( R_{11} \) is much larger than the other ones. Generally, the anisotropy can be introduced in two ways. The first one is based on Cholesky transformation when three velocity components of velocity induced by
vorton are rescaled. This way is not acceptable since it results in the loss of the divergence-free property. The second way is to dismiss the Cholesky transformation and to derive anisotropic turbulent spots. In this paper we present analytical solutions for such a spot obtained for the homogeneous turbulence.

2.4 Extension towards arbitrary anisotropy with retention of the continuity condition. Derivation of anisotropic vortons

The current work utilizes another approach for introducing the anisotropy into the turbulent spots which obeys continuity and allows to reproduce strong levels of anisotropy at the same time. The approach is basically a continuation of the vorton formulation described in [9]. The generation is performed in the coordinate system \((x, y, z)\) determined by principle axes of the Reynolds stresses. The Reynolds stresses in any other system \((x', y', z')\) are calculated as 
\[
R'_{ij} = E_{pq} R_{pq} E_{qj},
\]
where \(E_{ij}\) is the rotation matrix describing coordinate transformation between \((x, y, z)\) and \((x', y', z')\) axes system. Integral lengths in different systems can be found from the relation:
\[
L'_i(x', y', z') = \sum_{k=1}^{3} E_{ki} R_{kk} L_k(x, y, z)
\]

In [9], the vector potential is scaled by a function with spherical symmetry which in case of the spectrum of decaying turbulence gives an analytic expression:
\[
A(x, y, z) = C e^{-\frac{1}{2} k^2 r^2} \gamma
\]
Note that also other spectra \(E(k)\) could be used in principle. This would result in different shapes of the inner velocity distribution. For the sake of simplicity and because it yields reasonably simple formulas, the current work has been restricted to this spectrum.

The spherical symmetry of \(A(x, y, z)\) is the reason for the isotropy of turbulence generated using these vortons. At this level, anisotropy can be introduced by stretching the coordinates individually in a similar manner as it was done in [14], i.e.
\[
x \rightarrow x/\sigma_x \quad y \rightarrow y/\sigma_y \quad z \rightarrow z/\sigma_z.
\]

With this, the vector potential and velocity induced by vorton are now written as:
\[
A = \exp \left[ -\frac{1}{2} \left( \frac{x^2}{\sigma_x^2} + \frac{y^2}{\sigma_y^2} + \frac{z^2}{\sigma_z^2} \right) \right] \left( \begin{array}{c} x y \gamma_x \\ y z \gamma_y \\ z x \gamma_z \end{array} \right)
\]
\[
u = \exp \left[ -\frac{1}{2} \left( \frac{x^2}{\sigma_x^2} + \frac{y^2}{\sigma_y^2} + \frac{z^2}{\sigma_z^2} \right) \right] \left( \begin{array}{c} \frac{2 y}{\sigma_x} - \frac{2 z}{\sigma_y} \\ \frac{2 z}{\sigma_x} - \frac{2 x}{\sigma_y} \\ \frac{2 x}{\sigma_z} - \frac{2 y}{\sigma_z} \end{array} \right) \left( \begin{array}{c} x y \\ x z \\ y z \end{array} \right)
\]
Note that the multiplication with the coordinates is introduced to make the resulting Reynolds stress tensor a diagonal tensor which is identified with the diagonal matrix of eigenvalues from...
a principal component analysis of the prescribed Reynolds stress tensor. By aligning the $x$, $y$, $z$-directions of the vorton with the principal directions of the Reynolds stress tensor, arbitrary anisotropic Reynolds stresses can be reproduced. The vorton sizes $\sigma_x$, $\sigma_y$ and $\sigma_z$ and strength vector components $\gamma_x$, $\gamma_y$ and $\gamma_z$ are free parameters of the vorton and can be used to match the prescribed Reynolds stresses and integral length scales.

2.5 Statistical properties of anisotropic vortons

Statistical properties can analytically be derived for homogeneous turbulence. We consider the set of fully uncorrelated vortons, i.e. $\gamma_{ik}\gamma_{jm} = 0$ for each pairs of $k-th$ and $m-th$ vortons with strength components $i$ and $j$. Then the Reynolds stress $R_{ij}$ of the total field is equal to the sum of Reynolds stresses produced by each vorton

$$R_{ij} = u_i u_j = \sum_{k=1}^{n_s} u_{ik} u_{jk}$$

Without loss of generality we set the magnitude of the strength to be unit, i.e. $|\pm \gamma| = 1$. Then the expectation of the Reynolds stress $R_{ij}$ at the point $(0,0,0)$ is

$$R_{ij} = c \int \int \int u_i(\gamma, x) u_j(\gamma, x) P(x) dV$$

where $P(x)$ is the probability density function of the event that the vorton is placed at the point $x$. For the uniform distribution $P(x) = n_s/V = c$ is the vorton density. If the computational domain becomes infinite $n_s$ should increase, so that the vorton density remains constant:

$$R_{ij} = c \int \int \int u_i u_j dxdydz$$

(8)

Substitution of velocity, induced by anisotropic vorton (7), in (8) results in a simple formula

$$R = \frac{c^{3/2}}{4} \begin{pmatrix}
\sigma_x (\gamma_y \sigma_y^2 - \gamma_z \sigma_z^2)^2 & 0 & 0 \\
\sigma_y (\gamma_z \sigma_z^2 - \gamma_x \sigma_x^2)^2 & \sigma_y (\gamma_x \sigma_x^2 - \gamma_z \sigma_z^2)^2 & 0 \\
0 & \sigma_z (\gamma_x \sigma_x^2 - \gamma_y \sigma_y^2)^2 & \sigma_z (\gamma_y \sigma_y^2 - \gamma_x \sigma_x^2)^2
\end{pmatrix}$$

(9)

2.6 Determination of anisotropic vorton parameters $\sigma$ and $\gamma$.

Integration of autocorrelation functions reveals a simple and clear interpretation of stretching parameters $\sigma_i$:

$$L_x^2 = \int_0^\infty \rho_{xx}(\eta_x, 0, 0) d\eta_x = \sqrt{\pi} \sigma_x$$

$$L_y^2 = \int_0^\infty \rho_{yy}(0, \eta_y, 0) d\eta_y = \sqrt{\pi} \sigma_y$$

$$L_z^2 = \int_0^\infty \rho_{zz}(0, 0, \eta_z) d\eta_z = \sqrt{\pi} \sigma_z$$
Therefore, the parameters $\sigma_i$ are uniquely determined from the last formulas $\sigma_i = L_i / \sqrt{\pi}$. The vorton strength vector $\gamma$ is found from the condition for Reynolds stresses:

$$
\begin{align*}
\gamma_y \sigma_y^2 - \gamma_z \sigma_z^2 &= \pm 2 \sqrt{c^{-1} R_{11} \frac{L_y L_z}{L_x}} \\
\gamma_z \sigma_z^2 - \gamma_x \sigma_x^2 &= \pm 2 \sqrt{c^{-1} R_{22} \frac{L_x L_z}{L_y}} \\
\gamma_x \sigma_x^2 - \gamma_y \sigma_y^2 &= \pm 2 \sqrt{c^{-1} R_{33} \frac{L_x L_y}{L_z}}
\end{align*}
$$

(10)

where the upper index in $L$ is omitted for the sake of brevity. Since the determinant of (10) is zero, a solution of the system (10) is only possible if the following condition is satisfied:

$$
\pm \sqrt{R_{11} \frac{L_y L_z}{L_x}} \pm \sqrt{R_{33} \frac{L_x L_y}{L_z}} = \pm \sqrt{R_{22} \frac{L_x L_y}{L_z}}
$$

or

$$
L_y = \frac{\pm L_x L_z \sqrt{R_{22}}}{\pm L_z \sqrt{R_{11}} + \pm L_x \sqrt{R_{33}}}
$$

The signs before different terms are independent of each other. Therefore, the integral lengths can not be arbitrary. If two length scales $L_x$ and $L_y$ are prescribed the remaining length should satisfy the conditions above. Particularly, this solution is wrong for the isotropic turbulence since, if $R_{22} = R_{33} = R_{11}$ and $L_x = L_z = L$, the third length is $L_y = L/2$ although all integral lengths based on longitudinal autocorrelation functions should be equal.

This limitation can be overcome by mix of two statistically independent velocity fields $u_1$ and $u_2$ generated using the algorithm described above. While the derivation is straightforward, this has not yet been utilized in the course of the current work.

### 2.6.1 Difference with SEM [14].

At this stage, the following differences with the SEM [14] can be pointed out:

- The present method is based on simple analytic function (7) which has smooth derivatives of arbitrary order. As a result the divergence free condition is satisfied everywhere within the computational domain.
- The most serious advantage is that, three integral lengths $L_i$ can be explicitly prescribed. For that, the exact analytic relations were derived to express the vorton sizes $\sigma_i$ through $L_i$.

### 2.7 Application to inhomogeneous case

The anisotropic vortons derived above and described by simple formula (7) were obtained for the homogeneous turbulence, i.e. all statistical moments are invariant with respect to translation of the reference system. Only three integral lengths scales can be specified. The autocorrelation functions cannot be specified since they are predetermined and correspond to the spectrum of
decaying turbulence. The most serious limitation of this solution is the negligence of inhomogeneity. This is due to the fact, that derivation of relations for inhomogeneous case leads to complicated nonlinear integral equations whose analytic solution is not possible. Their numerical solution is also a very laborious procedure. Instead of that we propose to use homogeneous solutions which locally neglect their interaction. With other words, at each point \( x, y, z \) we place vortons with \( \sigma_i(x, y, z) \) and \( \gamma_i(x, y, z) \) calculated using stresses and lengths specified at this point. Due to overlapping of vortons with different properties, resulting stresses and lengths have some deviation from the specified ones. This deviation depends on the rate of the spatial change of \( Re \) and \( L \) and sizes of vortons \( \sigma_i \). For instance, in boundary layers \( Re \) stresses have the strongest gradient in normal direction \( y \). However, as shown in the next section, it is possible to obtain satisfactory results since the overlapping is governed by \( \sigma_y \) which is very small near the wall.

3 IMPLEMENTATION

The Turbulent Spot method has been implemented into the open source fluid dynamics package OpenFOAM® [19, 4]. The software OpenFOAM® was chosen because of its open and modular architecture and because it is sufficiently sophisticated to apply it to almost all technically relevant flow problems.

During layout of the implementation, focus was put on easy and simple application. The Turbulent Spot method was thus implemented as a self-contained boundary condition class. As an input, only Reynolds stress and length scale distributions have to be prescribed. The population of turbulent structures is generated and maintained online during the simulation.

3.1 Algorithm

Vortons are treated in a lagrangian manner. They are continuously generated upstream of the inlet boundary and are then convected through the boundary surface. During this period, the induced velocities on the boundary faces are computed. When a vorton has passed through the inflow boundary move so far downstream, that its induced velocity on the inlet vanishes, it is deleted from the vorton population. To best fit into the unstructured grid framework of OpenFOAM®, the algorithm works on a per-face basis.

The first goal is to ensure a uniform coverage of the inlet surface with vortons. A volumetric concentration of vortons is described by the parameter \( c \):

\[
c = \frac{n_s V_v}{V} = \frac{n_s}{V} L_1 L_2 L_3 \tag{11}
\]

with the number of vortons \( n_s \) covering a space volume \( V \) and the virtual volume of a vorton expressed by the product of its integral length scales \( V_v = L_1 L_2 L_3 \).

The value \( c \) is an external parameter for the generation procedure. It does not influence the generated statistics directly, i.e. after a sufficiently long averaging time the generated Reynolds stresses and length scales are equal, independent from the value of \( c \). A value much lower than unity \( c \ll 1 \) leads to intermittency. Large values lead to a growing number of vortons present per timestep and thus to an increase in computational effort and memory requirement. For the subsequent simulations, usually values of \( c = 2 \) have been used. This avoids intermittency by ensuring some reasonable amount of overlapping between neighboring vortons.
The uniform distribution of vortons in space is achieved by placing them in the swept volume of the face under consideration. Therefore, vortons are queued up with random face-normal distances, one vorton at each distance level. Each vorton is also randomly shifted within the plane parallel to its associated face, but only within the bounds of the face.

Each point in the swept volume shall have an equal probability for being chosen, which is related to the concentration parameter $c$

$$P(\vec{x}) = \frac{c}{V_v}$$  \hspace{1cm} (12)

The expected distance between subsequent vortons can be derived as

$$\langle \Delta v \rangle = \frac{V_v}{cA_f} = \frac{L_1L_2L_3}{cA_f}$$  \hspace{1cm} (13)

with $A_f$ being the area of the face under consideration. The vorton queue upstream of each face is continuously replenished so that all vortons which will affect faces during the current timestep are contained.

4 RESULTS

4.1 Channel Flow

The described method is applied to a generic channel flow test case. Channel flows essentially comprise two turbulent boundary layers at the top and bottom wall. The turbulence is anisotropic in the whole domain and isotropy is only approximately recovered in a small region near the channel mid plane. Compared to more complex flow cases, where the turbulence evolution downstream of the inlet might be quickly dominated e.g. by complicated geometry or other forcing, the turbulence in the channel is only caused by the presence of the flat wall and develops just very slowly naturally. These circumstances make the channel flow a quite challenging test case for turbulent inflow generation.

Channels with three Reynolds numbers have been simulated for which DNS results are available [11]. Parameters of all simulations are summarized in table 1.

4.1.1 Simulations with Cyclic Boundary Conditions

Conventionally, channel flow simulations are carried out without inflow turbulence generation using cyclic boundary conditions in all directions including the bulk flow direction and a mean pressure gradient for adjusting the bulk flow velocity. Such simulations have been done in advance of the inflow generator simulations to prove the mesh and numerical settings.

The results of the cyclic channel simulations suggest that the resolution and numerical settings are sufficient. The friction coefficients for all Reynolds numbers are plotted in figure 2 and the Reynolds stresses are shown in figure 1. All profiles are considered to be reasonably close to the reference values.

4.1.2 Inflow Generator Simulations

Once the numerical settings are validated, the cyclic boundaries in axial direction have been replaced by a velocity inlet/pressure outlet pair. In addition, the length of the channel was
Reynolds number \( \delta \) Domain size Resolution Grid size
\( \text{Re}_\tau \) \( H/2 \) \( L \times H \times W \) \( \Delta^+_x \times \Delta^+_y \times \Delta^+_z \) \( n_x \times n_y \times n_z \)
180 1 cyclic BCs 6 \times 2 \times 4.2 20 \times 2 \times 10 54 \times 48 \times 75
inflow BC 12 \times 2 \times 4.2 20 \times 2 \times 10 108 \times 48 \times 75
395 1 cyclic BCs 6 \times 2 \times 4.2 20 \times 2 \times 10 118 \times 64 \times 165
inflow BC 12 \times 2 \times 4.2 20 \times 2 \times 10 237 \times 64 \times 165
590 1 cyclic BCs 6 \times 2 \times 4.2 20 \times 2 \times 10 177 \times 75 \times 247
inflow BC 12 \times 2 \times 4.2 20 \times 2 \times 10 354 \times 75 \times 247

Table 1: Parameters of the channel flow simulations

increased and the pressure gradient forcing removed, because it becomes unnecessary. The parameters of these simulations are contained in table 1 as well.

<table>
<thead>
<tr>
<th>Reynolds number</th>
<th>turbulent structure type</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \text{Re}_\tau )</td>
<td></td>
</tr>
<tr>
<td>180</td>
<td>anisotropic vorton</td>
</tr>
<tr>
<td>395</td>
<td>hat spot</td>
</tr>
<tr>
<td></td>
<td>isotropic vorton</td>
</tr>
<tr>
<td></td>
<td>anisotropic vorton, ( R_{uv} = 0 )</td>
</tr>
<tr>
<td></td>
<td>DSRFG turbulence (“Modal”) [2]</td>
</tr>
<tr>
<td>590</td>
<td>anisotropic vorton</td>
</tr>
</tbody>
</table>

Table 2: Inflow channel simulations: computed variants

The reynolds number \( \text{Re}_\tau \) has been varied for simulations with the anisotropic vorton and for the middle Reynolds number \( \text{Re}_\tau = 395 \) only, the influence of the turbulent structure was evaluated.

The friction coefficient \( c_f = \tau_w / \frac{1}{2} \rho u_m^2 \) can be regarded as a measure of the reality level of the generated turbulence. As visible in figure 2, the friction coefficient vs. axial length should ideally be a horizontal curve. Figure 3a shows the friction coefficient evolution along the channel at \( \text{Re}_\tau = 395 \) for different turbulent structure types. Again, there is an initial deviation from expected value. It decreases with running length and is identified as the adaption distance. This distance depends on the type of structure used. The most primitive hat spot type needs the longest time to morph into real turbulence. A better behavior is obtained with the vorton types. With these structures, the adaption distance is about one channel height \( H \) shorter. The anisotropic vorton simulations have been carried out in two variants: with the full Reynolds stress tensor and with a diagonal-only tensor (\( R_{uv} = 0 \)).

In the former case, since the generated primary fluctuation field is always diagonal, the off-diagonals are produced by aligning the vortons with the principal axes of the prescribed Reynolds stress tensor. Since the longitudinal length scale is large compared to the lateral ones, an oblique orientation of the vortons leads to increased smoothing of the reproduced Reynolds stress profile (compare section 2.7). Also, oblique structures intersect with the wall and thus
create additional parasitic pressure noise. Thus in summary, aligning the structures by omitting $R_{uv}$ produces better results.

In figure 4, the friction coefficient for all three considered $Re_\tau$ is shown, normalized by its respective asymptotic value. This shall illustrate the sensitivity of the synthetic turbulence to the Reynolds number. The figure indicates that the behavior is quite comparable, if the Reynolds number is sufficiently large. Then the adaption length has the same length in terms of dimensionless wall units. But if the Reynolds number is sufficiently low, the artificial turbulence does not survive. This is at least the case for the near-transition case of $Re_\tau = 180$.

A big advantage of the divergence-free vorton structures over continuity-breaching structures

Figure 1: Cyclic channel flows: Reynolds stresses, a) $Re_\tau = 180$, b) $Re_\tau = 395$, c) $Re_\tau = 590$

Figure 2: Cyclic channel flows: friction coefficients
Figure 3: Inflow channel simulations at $Re_\tau = 395$: comparison of a) friction coefficient b) pressure fluctuations vs. axial distance for different types of turbulent structures.

like hat spots again becomes obvious from figure 3b: the artificial pressure noise is greatly reduced. While in the case of hat spots continuity is never fulfilled, isotropic vortons fulfill it only in the case of isotropy of the Reynolds stresses. Anisotropic vortons always fulfill continuity in the analytical case \(^1\). This is reflected by the hierarchy of curves in figure 3b. The largest artificial pressure noise is found for hat spots. That for anisotropic vortons is minimum while isotropic vortons are found in between. The difference between hat spots and anisotropic vortons is more than one order of magnitude.

5 SUMMARY

A novel type of turbulent spot has been derived. It allows fulfillment of the continuity constraint not only for isotropic Reynolds stresses but also for arbitrarily anisotropic ones.

The behavior of this structure has been demonstrated on a channel flow test case. Both case studies confirm that 1) the adaption length is positively influenced by application of vortons instead of simplicity-motivated velocity distributions like hat spots and 2) by being able to obey continuity always, the artificial pressure noise is greatly reduced.

6 LITERATURE

References


\(^1\)When the velocity distribution of an anisotropic vorton is imposed on a discrete mesh, residual continuity errors arise again
Figure 4: Inflow channel simulations: comparison of friction coefficient vs. axial distance for different Reynolds numbers (using the anisotropic vorton)


GRID GENERATION FOR WALL-MODELLED LES OF SHIP HYDRODYNAMICS IN MODEL SCALE

M. LIEFVENDAHL*, M. JOHANSSON* AND M. QUAS*

*Swedish Defense Research Agency (FOI)
164 90 Stockholm, Sweden

e-mail: mattias.liefvendahl@foi.se, web page: http://www.foi.se

Key words: Wall-Modelled Large-Eddy Simulation, Ship Hydrodynamics, Grid Generation

Abstract. An unstructured grid generation approach for wall-modelled LES is proposed. The applicability of the approach is demonstrated for the simulation of the flow around an axisymmetric body, at Re-number 5.48 · 10^6, which is representable of model scale ship hydrodynamics. A numerical trip wire must be employed to induce resolved fluctuations in the simulated boundary layer. The predictive accuracy of the simulation technique is evaluated for the flow around the axisymmetric body, and for the computation of a turbulent boundary layer on a flat plate. For the flat plate, comparison is made with results from direct numerical simulation. For the axisymmetric body, results from wall-modelled LES, RANS and experimental measurements are compared.

1 INTRODUCTION

In the application of large-eddy simulation (LES) to ship hydrodynamics, the treatment of the turbulent boundary layer (TBL) along the hull is often of critical importance. The resulting computational cost is strongly dependent on the choice of the modelling technique for the TBL, and on the Reynolds (Re)-number of the simulated case. A fully developed TBL, as for instance along a ship hull, can be characterized by its thickness $\delta$, and its viscous length scale, $\delta_v = (\rho \nu^2 / \tau_w)^{1/2} \ll \delta$, given in terms of the density $\rho$, the kinematic viscosity $\nu$, and the wall-shear stress $\tau_w$. The most straightforward application of LES is to resolve the energetic flow structures in the inner part of the TBL, of size $\sim \delta_v$, which is referred to as wall-resolved LES (WRLES). If the simulation resolves the larger flow structures in the bulk of the TBL, but employs special modelling of the effects of the very near-wall flow on these larger flow structures, then the approach is referred to as wall-modelled LES (WMLES). In section 2, this classification is elaborated on, and implications for the generation of the computational grid for WMLES are discussed.

The present paper is concerned with WMLES applied to ship model scale simulations, which implies a range of hull-length based Re-numbers from $10^6$ to $10^7$. It is estimated [13], that WMLES would, at least, require in the order of $20 \cdot 10^6$ grid points for a simulation of the hull boundary layer. The corresponding estimate for WRLES is $\sim 10^9$ grid points. Recently
the first WRLES simulation of a ship in model scale was reported, [20]. Despite the fact that WMLES would require significantly less computational resources, there is a striking lack in the literature of reported ship simulations of this type. There are many simulations using DES or hybrid RANS/LES methods, see e.g. [29, 9, 12, 15], but for these the entire TBL is treated in the RANS sense. For the past three decades, there has been a range of more fundamental papers on WMLES, where channel flow has been the dominating simulation case, see e.g. [22, 18] and the review of wall models in [24]. An example of an application to a geometry more complicated than an orthogonal box (as for the channel flow) is the airfoil simulation reported in [2].

The main contribution of the present paper is to describe and demonstrate an unstructured grid generation approach for WMLES, which is suitable for model scale ship hydrodynamics. The guiding principle is to adapt the grid refinement level to $\delta$, which is estimated a priori. In Section 2, estimates of the TBL length scales are first reviewed, and then the grid generation approach is described by its application to the simulation of the flow around an axisymmetric body. Results from two simulation cases are also included to in the paper to provide information concerning the predictive accuracy of WMLES. The first case, see Section 4, consists of a TBL developing over a flat plate. Results from WMLES are compared with DNS (direct numerical simulation) for validation. The second case, presented in Section 5, is the flow around the axisymmetric body, for which the grid generation approach was presented. WMLES results are compared with experimental measurements of skin friction and the mean velocity distribution in the stern region. A brief review is included in Section 3, of the turbulence modelling and numerical methods which are employed in the two simulations cases.

2 GRID REQUIREMENT ESTIMATES AND IMPLICATIONS FOR GRID GENERATION

A convenient starting point for discussing grid generation for WMLES is to estimate the number of grid points required to represent the relevant flow scales. In an early paper [3], estimates were derived for WMLES, as well as for DNS and WRLES. In subsequent papers, [27, 4, 23], the estimates, and the employed expressions for TBL thickness and friction coefficient, were refined. The relevant material is reviewed here, before the practical grid generation is described.

Consider a (material) surface $S$ over which there is an attached TBL (no-slip boundary condition and TBL on one side of $S$). Locally, at a point $x$ on the surface, there is TBL thickness $\delta(x)$ and a viscous length scale $\delta_v(x)$. Estimates for these can be obtained by correlating power law expressions to available results from DNS and experiments, see e.g. [23].

2.1 Length-scale estimates for a TBL over a flat plate

For a zero pressure gradient flat plate TBL (ZPGFPTBL), the following relations, or power laws, were used in [23].

$$\frac{\delta(x)}{x} = \alpha_1 Re_x^{-\beta_1} \quad c_f = \alpha_2 Re_x^{-\beta_2} \quad \delta_v = \frac{\nu}{V_0} \left( \frac{2}{c_f} \right)^{1/2}$$  \hspace{1cm} (1)

Here $\delta$ is the wall distance at which the mean velocity is 99% of the free-stream velocity, $x$ is the distance from the leading edge of the flat plate, $V_0$ is the free-stream velocity, $c_f = 2\tau_w/(\rho V_0^2)$
is the friction coefficient, and $Re_x = xV_0/\nu$. The model coefficient values,
\[
\alpha_1 = 0.1222, \quad \beta_1 = 0.1372, \quad \alpha_2 = 0.0283, \quad \beta_2 = 0.1540,
\]
were obtained by correlation with experimental data [21]. A comparison with a compilation of experimental data [17], and DNS [25, 26] showed that the expressions (1) are quite accurate in the range, $10^6 < Re_x < 5 \cdot 10^7$.

As an example, we apply the expressions for $\delta$ and $\delta_\nu$ to the TBL mid-ships in a typical model scale experiment. Assuming a model length of $L = 5$ m, a towing speed $V_0 = 2$ m/s, and a kinematic viscosity $\nu = 10^{-6}$ m$^2$/s, gives $Re_x = 5 \cdot 10^6$. For these parameters, the expressions (1) give $\delta = 37$ mm, and $\delta_\nu = 14 \mu$m, i.e. $\delta/\delta_\nu \approx 2700$. It is quite clear that a great gain in computational cost can be achieved if the grid is constructed to resolve $\delta$ and not $\delta_\nu$, i.e. by using WMLES instead of WRLES.

2.2 Grid estimates for WMLES

The fundamental assumption in the present paper is that the grid for WMLES should be adapted to $\delta$. This is translated to a requirement of a fixed number of grid points $n_0$ in a $\delta^3$-cube of the TBL. Values in the range, $500 < n_0 < 10^4$ have been reported in the literature [3, 27, 5]. See also sections 4 and 5 below for practical demonstrations grid resolution and achieved predictive accuracy. This means that we have a local density of grid points, $\rho_N(\mathbf{x}) = n_0/\delta(\mathbf{x})^3$, and we obtain the total number of grid points $N$ by integrating this over the TBL at the surface $S$.

\[
N = \int_S \int_0^{\delta(\mathbf{x})} \rho_N(\mathbf{x}) \, dy \, dS = \int_S \frac{n_0}{\delta(\mathbf{x})^2} \, dS
\]

Here $y$ is the local wall-normal coordinate, with $y = 0$ at the wall. A number of observations concerning expression (3) can be made:

1. The grid estimates, [3, 27, 4, 23], were obtained by inserting a power law for $\delta$ which is valid for a FPZPGTBL. It is referred to the original papers for the derivation, and the resulting grid estimates.

2. The integral (3) is only valid for a fully developed TBL. Its initial section, comprising the laminar boundary layer and its transition to turbulence, must be handled separately.

3. The integral (3) only covers the TBL-region. Grid estimates for the region of the flow away from the wall/TBL must be handled separately.

The practical implications of these observations are discussed in the next section, and in connection with the two simulation cases included in the paper.

2.3 Grid generation

A grid generation method for WMLES is demonstrated by applying it for the construction of a computational grid for the computation of the external flow around the axisymmetric body shown in Fig. 1. The body is the unappended bare hull geometry of the generic submarine design proposed in [10, 11], see also [1]. The grid is to be constructed for flow at a length-based
Re-number, $LV_0/\nu = 5.48 \cdot 10^{6}$, see Table 2 for the complete set of parameters of the simulated case, and Section 5 for a description of the simulation results.

The boundary layer over the hull starts at the stagnation point on the bow and develops first as laminar, affected by a favorable pressure gradient. The pressure coefficient on the hull is also shown in Fig. 1. Generally, a favorable pressure gradient has a stabilizing effect, whereas an adverse pressure gradient supports transition. The simulated case corresponds to wind tunnel measurements, see section 5, in which a trip wire was included to fix the location of the boundary layer transition. The location of the trip wire, $x/L = 0.05$, is shown in Fig. 1. Here the $x$-coordinate is directed in the main flow direction, and $x = 0$ at the stagnation point. Downstream of the trip wire, the boundary layer is turbulent and it is attached to and develops along the hull. At the stern, the TBL is affected by an adverse pressure gradient which leads to a rapid thickening of the TBL. There is no separation of the mean flow at the stern. As can be seen in Fig. 1, the pressure gradient is relatively weak downstream of the location of the trip wire. In the WMLES computation, a numerical trip wire is employed at the same location as the trip wire used in the experiments.

The objective is now to construct an unstructured grid which is adapted to the local thickness of the TBL. From now on the discussion is in terms of a grid for finite volume discretization of the governing equations over the grid cells. The general approach however is applicable also in other numerical frameworks such as the finite element method. In the chosen approach the following three steps are carried out in order to create the complete grid of the computational domain.

1. The hull surface is grid is first created by triangulating the surface.
2. A certain number of layers of prismatic cells are extruded from the surface triangulation.
3. A tetrahedral grid is generated for the domain outside of the layers of prismatic cells.

The adaption of the cell size to the local TBL thickness is primarily done at the first stage, the surface meshing. The assumed variation of the TBL thickness is given by the following expression.

\[
\delta(x) = \begin{cases} 
\delta_0, & x < x_1 \\
\alpha_1 x \text{Re}^{-\beta_1}, & x \geq x_1 
\end{cases}
\]  

(4)

Here \(x_1\) is chosen at a suitable location just downstream of the trip wire, at \(x/L \approx 0.06\), and \(\delta_0\) is chosen to have continuity of the function \(\delta(x)\) at \(x_1\). This is next translated to a recommended cell/triangle edge length \(\Delta l\) by the relation,

\[
\left(\frac{\delta}{\Delta l}\right)^3 = n_0,
\]

(5)

which corresponds to \(n_0\) cubical cells in the (local) \(\delta^3\)-cube. It would be possible to include a constant factor in equation (5) to account for the difference in using tet/prism cells, and not cubical cells. This would however exactly correspond simply to a scaling of \(n_0\).

Now parameters are chosen for the grid generation for the simulation for which results are presented in Section 5. The values \(\delta_0 = 4\) mm, and \(n_0 = 500\), are chosen. This leads to a recommended edge length of, \(\Delta l \approx 0.5\) mm, for the triangles of the grid over the bow. The gradual increase of \(\delta(x)\) downstream of \(x_1\) leads to a thickness of \(\delta \approx 15\) mm, at the end of the parallel section of the hull, which corresponds to \(\Delta l \approx 2\) mm, at this station. The total number of triangles in the resulting surface mesh is approximately \(1.25 \cdot 10^6\).

The second step is to extrude prismatic cells from the surface triangulation. The cells are constructed with a height aspect ratio of, \(\Delta h/\Delta l \approx 0.5\), and 16 layers are created without any expansion ratio away from the wall. This leads to a prismatic TBL grid covering approximately the full width \(\delta\). The total cell count of the TBL grid is \(20.5 \cdot 10^6\) prisms.

The third and last step is to construct a tetrahedral grid for the remainder of the computational domain. In the present case, the focus of the investigation is the TBL. Hence, no special regions with increased grid density are needed, and it is possible to rapidly coarsen the mesh away from the TBL. The generated grid contains \(5.7 \cdot 10^6\) tetrahedral cells, leading to a total grid cell count of \(26.3 \cdot 10^6\).

3 CFD METHODS

The turbulence modelling and numerical methods employed in the two simulations, presented in Sections 4 and 5 respectively, are briefly summarized in this section, and references are given to complete descriptions.

3.1 Large-eddy simulation

Spatial filtering of the incompressible Navier-Stokes equations, and neglecting commutation errors, lead to the fundamental LES equations, see e.g. [24]:

\[
\frac{\partial \mathbf{v}}{\partial t} + \nabla \cdot (\mathbf{v} \otimes \mathbf{v}) = -\frac{1}{\rho} \nabla p + \nabla \cdot (\mathbf{S} - \mathbf{B}) \\
\nabla \cdot \mathbf{v} = 0
\]

(6)
Here $\mathbf{v}$ is the (filtered) velocity field, $\rho$ the density, $p$ the pressure, $\mathbf{S} = 2\nu \mathbf{D}$ the viscous strain tensor, $\mathbf{D} = (\nabla \mathbf{v} + \nabla \mathbf{v}^T)/2$ the rate-of-strain tensor and $\nu$ the kinematic viscosity. The term in equation (7) arising from the filtering, is the subgrid stress tensor, $\mathbf{B} = \mathbf{v} \otimes \mathbf{v} - \mathbf{v} \otimes \mathbf{v}$. Here, variables with overbar denotes filtered quantities. Below however, the overbar is dropped in order to simplify the notation. It is referred to [24] as a general reference for LES and [8], and the references therein, for LES applied to problems in naval hydrodynamics.

The LES modelling consists of providing an expression for $\mathbf{B}$. In the simulations reported below, the wall-adapting local eddy viscosity (WALE) model, by Nicoud and Ducros [19], is employed. This is an eddy-viscosity model, which means that the subgrid stress is computed as $\mathbf{B} = -2\nu_k \mathbf{D}$, where $\nu_k$ is the subgrid viscosity. In the WALE model, $\nu_k$ is determined using an expression including the velocity gradient tensor.

For the flow next to the wall, special subgrid modelling must be used in the form of a so-called wall model. The one employed in the simulations of this paper uses the cell-centered velocity in cells adjacent to the wall, and the expression for the law-of-the wall by Spalding [28], to enforce the correct wall-shear stress in each time step. It is referred to [6, 7, 14] for complete description of this wall model.

### 3.2 The finite volume discretization

The solver which is used is implemented using the open source software package OpenFOAM\(^1\), which provides an object-oriented library, based on the finite-volume method, specifically designed for CFD, see [30] for the original description of the structure of this software design. The LES subgrid models have been implemented inhouse at FOI.

The discretization of the governing flow equations relies on storage of the unknown flow variables in the cell-center positions in the computational grid. The algorithm supports arbitrary polyhedral cells and the grid is treated as unstructured. The approximations involved are of second-order accuracy, except for flux limiting for the convective term, which reduces locally the formal order of accuracy near sharp gradients. The momentum equation is treated in a segregated manner, solving sequentially the three components of the momentum equations in a loop within each time step.

The simulations are time resolved and a second order backward differencing scheme is used for the time advancement of the components of the momentum equation. A domain decomposition technique, applied to the grid, in combination with an efficient MPI-implementation is used for running on parallel computers.

### 4 FLAT PLATE TURBULENT BOUNDARY LAYER

The turbulent boundary layer over a flat plate is simulated using WMLES at a length-based Re = 2.47 · 10^6. The predictive accuracy is evaluated by comparison with DNS-results. For the WMLES, a block-structured grid with uniform hexahedral cells is used in the rectangular simulation domain. Hence the unstructured grid generation approach described above is not used for this case. Nevertheless, the simulated case provides useful information concerning the predictive accuracy of WMLES, with a reasonable grid resolution relative to the TBL thickness. The case is especially useful because of the availability of DNS-data for validation.

\(^1\)www.openfoam.com
### Table 1: Physical parameters for the flat plate simulation case.

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Notation</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plate length</td>
<td>L</td>
<td>2.002</td>
<td>m</td>
</tr>
<tr>
<td>Channel height</td>
<td>h</td>
<td>0.133</td>
<td>m</td>
</tr>
<tr>
<td>Plate width</td>
<td>b</td>
<td>0.200</td>
<td>m</td>
</tr>
<tr>
<td>Kinematic viscosity</td>
<td>ν</td>
<td>1.65\times10^{-5}</td>
<td>m$^2$/s</td>
</tr>
<tr>
<td>Free-stream velocity</td>
<td>$V_0$</td>
<td>20.4</td>
<td>m/s</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>$Re = V_0 L/ν$</td>
<td>2.47\times10^6</td>
<td>- - -</td>
</tr>
</tbody>
</table>

### 4.1 Parameters, pre-processing and simulation

The physical parameters of the flat-plate case are summarized in Table 1. The flow takes place in a rectangular domain,

$$(x, y, z) \in [0, L] \times [0, h] \times [0, b],$$

where $L$ is the length of the plate, $b$ is the width of the plate, $h$ is the height of the simulation domain (channel), and the main flow is in the $x$-direction. A uniform inflow velocity, $V_0$, is prescribed at the inflow at $x = 0$. The plate is the bottom of the channel, $y = 0$, at which no-slip boundary conditions are applied. At the top boundary, $y = h$, a slip boundary condition is applied, and in the $z$-direction periodic boundary conditions are employed. A simple orthogonal grid was used, with $760 \times 160 \times 100 = 12,160,000$ uniform hexahedral cells.

One WMLES computation was carried out using the WALE subgrid model. It is referred to Section 3 for a description of the wall model and the numerical methods. In order to induce resolved fluctuations in the boundary layer, a numerical trip wire is applied on the plate, next to the inflow. This is affecting the flow through a random volume force applied in a strip of cells next to the plate.

### 4.2 Results for the flat plate

The flow over the flat plate, simulated by WMLES, is illustrated in Fig. 2, which shows the instantaneous axial velocity and the wall shear stress on the plate. At two locations, one near the inflow and one near the outflow, the grid resolution relative to the length scales of the flow structures is also illustrated. For this simulation, naturally, the grid resolution relative to the TBL is increasing because of the uniform cell size and the growing thickness of the TBL along the plate.

The development of the mean friction coefficient and the momentum thickness, $θ$, of the TBL are shown in Fig. 3. The WMLES-results are compared with the power law expression for $c_f$, see equation (1). In the absence of a pressure gradient, the relation, $c_f = 2dθ/dx$, holds and can be integrated to yield a power law for $θ$. Good agreement is seen between the WMLES and the power law. The “wiggles” in $c_f$ at, $x \approx 0.15$, is related to the numerical trip wire and the development of resolved fluctuations in the simulation. The $θ$-curve for WMLES is shifted in the $x$-direction relative to the power law. This is also explained by the initial development of the TBL. After this initial section, the WMLES-results demonstrate the corrected TBL-thickness growth.
Figure 2: The flat-plate TBL simulated by WMLES. On the bottom wall, the friction coefficient, $10^3 \cdot c_f$, is plotted, and on a vertical plane the normalized instantaneous velocity, $v_x/V_0$, is plotted. The above graph shows the whole channel. The two pictures below are zoomed-in at the locations marked by 'A' and 'B' respectively. In the below pictures, the grid is also shown on the bottom wall and the vertical plane.

Figure 3: Mean friction coefficient and momentum thickness of the flat plate TBL. Comparison of WMLES-results with the power laws (1). Above: Mean friction coefficient, $\langle c_f \rangle$, as a function of $x$. Below: TBL momentum thickness, $\theta$, as a function of $x$. 
Profiles of flow statistics are shown in Fig. 4. WMLES-results are compared with DNS, see [25], at a station corresponding to, $\theta V_0/\nu = 3.626$. For the WMLES-computation, this station is located at, $x = 1.76$ m, i.e. relatively near the outlet. The grid cell height is, $\Delta y = 0.8$ mm, which at this station corresponds to, $\Delta y^+ = \Delta y/\delta_\nu \approx 41$, based on the viscous length scale calculated by WMLES. The first cell center is thus located at, $y^+ \approx 20$, and the mean velocity is, $\langle v_x \rangle \approx 0.45 V_0$, in this point. Considering the level of grid resolution, the agreement of the profiles can be considered to be relatively good. The peak value of the turbulent shear stress is well predicted. The main discrepancy is that the turbulent stress profile decays too slowly at away from the wall. This is associated with large TBL flow structures which intermittently reaches out to a wall distance of 25-30 mm in the WMLES. Based on a boundary layer thickness of $\delta \approx 3$ mm, the grid resolution at this station is $\approx 5000$ cells in a $\delta^3$-cube. This can be considered as relatively high. The TBL prediction is however affected by the development of the TBL along the plate where the grid resolution relative to $\delta$ is lower.

5 FLOW AROUND AN AXISYMMETRIC BODY

The flow around an axisymmetric model in a wind-tunnel is simulated using WMLES at a length-based $\text{Re} = 5.48 \cdot 10^6$. The results are compared with measurements of the skin friction and the mean velocity at the stern and in the near wake. The grid generation approach described in the Section 2.3 was employed for the simulation.

5.1 Parameters, pre-processing and simulations

The physical parameters of the axisymmetric body simulation case are summarized in Table 2. This corresponds to experiments carried out in the DSTO low speed wind tunnel, [1]. A
Cartesian coordinate system is used with the main flow in the positive $x$-direction and, $x = 0$, at the bow of the body. The flow is simulated in a rectangular computational domain, occupying, $-1.25 \, \text{m} < x < 6.44 \, \text{m}$, and with cross-section, $1.92 \, \text{m} \times 1.92 \, \text{m}$. At the inlet, the constant flow velocity $V_0 = 60 \, \text{m/s}$, is prescribed.

### Table 2: Physical parameters for the axisymmetric body simulation case.

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Notation</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hull length</td>
<td>$L$</td>
<td>1.350</td>
<td>m</td>
</tr>
<tr>
<td>Hull diameter</td>
<td>$D$</td>
<td>0.185</td>
<td>m</td>
</tr>
<tr>
<td>Kinematic viscosity</td>
<td>$\nu$</td>
<td>$1.478 \cdot 10^{-5}$</td>
<td>m$^2$/s</td>
</tr>
<tr>
<td>Free-stream velocity</td>
<td>$V_0$</td>
<td>60.0</td>
<td>m/s</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>$Re = V_0L/\nu$</td>
<td>$5.48 \cdot 10^6$</td>
<td>- - -</td>
</tr>
</tbody>
</table>

One WMLES computation was carried out using the WALE subgrid model on a grid generated according to the procedure described in Section 2.3, with a total of $26 \cdot 10^6$ cells. To induce resolved fluctuations, a numerical trip wire was employed on the bow of the body, as described in Section 2.3. For comparison, the flow was also simulated using the $k-\omega$ SST turbulence model, [16], as implemented in the standard OpenFOAM-solver `simpleFoam`.

**Figure 5:** Visualization of the flow around the axisymmetric body simulated using WMLES. In the top half, an iso-surface of the second invariant of the velocity gradient is shown, illustrating the resolved fluctuations in the TBL. In the lower half, the normalized axial velocity, $v_x/V_0$, is shown on the center plane. The iso-surface is also colored by $v_x/V_0$, using the same color scale.

### 5.2 Results for the axisymmetric body

The flow around the axisymmetric body, simulated by WMLES, is illustrated in Fig. 5. There are resolved fluctuations in the TBL over the complete hull downstream of the trip wire. The skin friction along the body is plotted in Fig. 6. The measurement were obtained using the Preston tube method, see [1] and the references therein for more information. The qualitative behaviour of the three $c_f$-curves is quite similar. The results indicate an underprediction with $\approx 30\%$ for WMLES as compared to the measurements. This may be associated with a thinner TBL in the WMLES simulation. The cause for this may be related to the action of the trip wire or the mesh resolution relative to the TBL. This discrepancy does not however appear to have
a significant effect on the thickening of the TBL over the stern, and the formation of the wake, as discussed next.

Figure 6: Mean friction coefficient, \(\langle c_f \rangle\), as a function of \(x/L\) along the axisymmetric body. Comparison of results from WMLES, RANS and measurements, [1].

Figure 7: Comparison of profiles of the mean axial velocity, \(\langle v_x \rangle/V_0\), obtained by WMLES, RANS and measurements, [1], respectively. The top graph illustrates the three lines (black) along which the profiles have been extracted, at \(x/L = 0.9, 1.0\) and \(1.1\) respectively. The vertical axis in the three profile plots is the vertical coordinate \(y\) (with \(y = 0\) on the symmetry axis of the body).
The mean velocity at the stern and in the near wake is shown in Fig. 7. Mean velocity profiles are extracted at three stations, and results are compared from WMLES, RANS and measurements. Hot wire anemometry was used in the measurements, see [1] and the references therein for more information. At the first station, \( x/L = 0.9 \), when the adverse pressure gradient has just started to affect the TBL, there are noticeable differences between the profiles. The measurements cannot reach very close to the surface, and there also appear to be a slight displacement of the measured profile out from the boundary. The WMLES profile, at \( x/L = 0.9 \), is most likely too thin, which would correlate well with the observation concerning the underprediction of \( c_f \) above. Nevertheless, there is a quite good agreement between all three sets of results for the profiles at the stern, \( x/L = 1.0 \), and in the near wake, \( x/L = 1.1 \). This is an indication that WMLES may be robust for prediction of the flow away from walls, even if the TBL is not well resolved, \( n_0 = 500 \) for the present case, or very accurately represented.

6 CONCLUSIONS

An unstructured grid generation approach for WMLES, suitable for model scale ship hydrodynamics, have been described, see Section 2.3, and demonstrated for the simulation of the flow around an axisymmetric model, see Section 5. The main feature of the proposed approach is to adapt the grid to the local TBL thickness, which is estimated a priori. An essential component of the method is to employ a numerical trip wire in order to induce resolved fluctuations in the boundary layer. Aspects of the placement of the numerical trip and the requirements on the grid in this region are also discussed. There is potentially a great gain in computational cost if WMLES can be applied instead of WRLES. In the WMLES of the axisymmetric body, \( 26 \cdot 10^6 \) grid cells were used, whereas the it is estimated that WRLES for the same case would require in the order of \( 10^9 \) cells, [23].

The predictive accuracy of the overall WMLES approach is evaluated by its application to two simulation cases: (i) The turbulent boundary layer over a flat plat, and (ii) The flow around an axisymmetric body. For the flat plate, the results are validated by comparison with DNS. For the axisymmetric body, results are compared between WMLES, RANS and experimental measurements. Integral quantities, such as skin friction and TBL thickness, are included in the evaluation, as well as profiles of the mean velocity and the turbulent shear stress. The overall conclusion is that WMLES, with the proposed unstructured grid generation approach, is promising for model scale ship hydrodynamics. The predictive accuracy is relatively good, and the computational cost is well within the scope of many current medium scale high-performance computing systems.
REFERENCES


ON THE PREDICTION OF SHEAR-LAYER FLOWS WITH RANS AND SRS MODELS

G. Vaz\(^1\), F.S. Pereira\(^{1,2,3}\) and L. Eça\(^3\)

\(^1\)Maritime Research Institute Netherlands, Wageningen, the Netherlands
\(^2\)Texas A&M University, College Station, United States of America
\(^3\)Instituto Superior Técnico, Lisbon, Portugal

Key words: Reynolds-Averaged Navier-Stokes equations; Scale-Resolving Simulations; Modelling accuracy; Shear-layer flows; Cylinder flows.

Abstract. This study evaluates the ability of Reynolds-Averaged Navier-Stokes (RANS) and Scale-Resolving Simulations (SRS) models to predict turbulent shear-layer predominant (blunt-body) flows. The selected cases are the flows around a circular cylinder at \(Re = 3,900\) and \(140,000\), and past a rounded square prism at \(Re = 100,000\) and incidence angles of 0 and 45 degrees. These cases exhibit complex features making numerical predictions a challenge, in particular, for turbulence modelling: shear-layers (free, boundary and wake), laminar-turbulent transition, low to moderate Reynolds numbers, flow separation and unsteadiness. In this paper, the aforementioned cases are simulated employing isotropic and anisotropic RANS, Delayed Detached-Eddy Simulation (DDES), eXtra Large-Eddy Simulation (XLES), and Partially-Averaged Navier-Stokes (PANS) equations. The outcome confirms that traditional isotropic RANS are unable to accurately predict such flows, whereas SRS models can significantly reduce modelling errors. Furthermore, the results show that anisotropic RANS models are an interesting engineering option owing to its compromise between accuracy and cost. Nonetheless, an improvement of the modelling accuracy by both anisotropic RANS and SRS models is inevitably coupled with an increase of the numerical demands.

1 INTRODUCTION

Several flows with relevance to naval and maritime hydrodynamics are characterized by high Reynolds numbers (\(Re > 10^5\)), laminar-turbulent transition, turbulence, complex and/or blunt geometries that originate shear-layer flows, flow separation, and regions that might be dominated by unsteady phenomena. All these features contribute to hamper the accurate numerical representation of such flows, making turbulence modelling a critical aspect. Although traditional isotropic Reynolds-Averaged Navier-Stokes (RANS) equations are widespread in the maritime community and are relatively affordable for the computational resources available in this scientific field, this approach has a limited modelling accuracy in predicting the aforementioned flows.
Contrary to RANS, where all turbulence scales are modelled and averaged, Scale-Resolving Simulation (SRS) models are able to solve some of the turbulence scales, theoretically increasing the modelling accuracy. However, their correct application cannot be done without understanding their main principles, while their accurate numerical solution may be excessively demanding for some practical applications.

Within these SRS models we emphasize two types: 1) the so-called hybrid methods which combine a RANS approach in near-wall regions with a LES model in outer and detached regions (e.g. Detached-Eddy Simulation (DES), Delayed Detached-Eddy Simulation (DDES), Very Large-Eddy Simulation (VLES), Extra Large-Eddy Simulation (XLES)); 2) the so-called bridging models, which employ the same turbulence approach in the entire domain, whether or not the equations’ filter is constant in space and time (e.g. Partially-Averaged Navier-Stokes (PANS) equations and Partially-Integrated Transport Model (PITM)). Naturally, hybrid and bridging formulations are dependent on the physical resolution and on the quality of the underlying (RANS-based in some cases) turbulence model. In terms of turbulence modelling, several approaches have been developed to improve the turbulence models’ accuracy by exploiting, for instance, the non-linearity between the Reynolds stresses and the product of the eddy-viscosity and the strain-rate tensor, or the anisotropy of the turbulent field (e.g the lag model, $v^2 - f$ model, Explicit Algebraic Reynold-Stress Model (EARSM), Reynolds-Stress Model (RSM)). Additionally, whenever laminar-turbulent boundary-layer transition plays a role, and the application of SRS is rather of no use (if the models do not solve turbulence at all in the boundary layer) or too expensive (if the models try to solve the transition-relevant small scales in the boundary layer), modern RANS-based transition models might be an attractive solution (e.g. Local Correlation Transition Model (LCTM), $k - k_l - \omega$, Wilcox 2006).

In the last years substantial amount of work has been published by the authors in order to study, implement and verify some of these models, as well as validate their application for several types of blunt-body flows [3, 16–19, 21, 22, 24]. This paper summarizes some of the findings of these previous studies, and compares the performance of some of these new approaches for typical structures used in the maritime industry. The selected models are RANS supplemented with the isotropic $k - \omega$ Shear-Stress Transport (SST) model [10] with or without the Local Correlation Transition Model (LCTM) [9], RANS in combination with the Explicit-Algebraic Reynold-Stress Model (EARSM) of [2], DDES [5], XLES [7], and PANS [4]. These models are evaluated on four distinct test-cases: the flow around a circular cylinder at Reynolds numbers equal to 3,900 and 140,000, and the flow around a rounded square prism at $Re = 100,000$ and two angles of incidence (0 and 45 degrees)\(^1\). The reasoning behind the selection of these benchmark cases lies on the complexity of the flow physics, simplicity of gridding, and availability of detailed experimental measurements. All this makes these cases extremely useful to gain insight on the selected models, in order to prepare their application to more industrially relevant flows. Note that in these industrial cases, usually the additional geometrical details simplify/force the flow solution by fixing transition and separation zones. Therefore, such flows are not as complex in terms of flow topology and flow phenomena as these canonical test cases that present geometries with continuous smooth walls. Nonetheless, application of the selected models to industrial applications may become computationally expensive.

\(^1\)Work on a square prism at $Re = 22000$ is ongoing [12].
Detailed Verification and Validation studies have been done for some of the cases tackled here [3, 16–19, 21, 22]. In particular, the following numerical studies have been performed: effect of the computational domain size; influence of boundary conditions and third dimension size; consequences of grid layout; influence of iterative convergence; influence of simulation time and quantification of statistical uncertainty; spatial and temporal resolution and quantification of associated discretization uncertainty; influence of explicit turbulence filter. In this paper, we present only one particular set of all those numerical settings tested.

The paper is organized as follows. Section 2 summarizes briefly the major mathematical approaches employed in the current work. Section 3 describes the problems addressed and the numerical setup used. Thereafter, Section 4 presents and discusses the numerical results, while Section 5 presents the conclusions.

2 MATHEMATICAL MODELS

Consider the existence of an arbitrary filter (implicit or explicit), constant preserving, and commuting with spatial and temporal differentiation. The application of such a filtering operator, which decomposes any dependent quantity $\Phi$ into a resolved $\langle \Phi \rangle$ and a modelled $\phi$ component such that $\Phi = \langle \Phi \rangle + \phi$, to the continuity and momentum equations leads to

$$\frac{\partial \langle V_i \rangle}{\partial x_i} = 0,$$

(1)

$$\frac{D\langle V_i \rangle}{Dt} = -\frac{1}{\rho} \frac{\partial (P)}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \nu \left( \frac{\partial \langle V_i \rangle}{\partial x_j} + \frac{\partial \langle V_j \rangle}{\partial x_i} \right) \right] + \frac{1}{\rho} \frac{\partial}{\partial x_j} \left( \tau_{ij}(v_i, v_j) \right),$$

(2)

where the flow is assumed to be incompressible and single-phase. In the previous equations $x_i$ are the coordinates of a Cartesian coordinate system, $V_i$ are the Cartesian velocity components, $P$ is the pressure, $\rho$ is the fluid density, $\nu$ is the kinematic viscosity and $\tau_{ij}(v_i, v_j)$ is a tensor produced by the filtering process that contains the effect of the unresolved flow field $\phi$ on the resolved velocity components $\langle V_i \rangle$ as a diffusion-like term. In order to model $\tau_{ij}(v_i, v_j)$ the Boussinesq hypothesis is used,

$$\frac{\tau_{ij}(v_i, v_j)}{\rho} = 2\nu_t \langle S_{ij} \rangle - \frac{2}{3} k \delta_{ij},$$

(3)

where $\nu_t$ is the eddy-viscosity, $\langle S_{ij} \rangle$ the resolved strain-rate tensor, $k$ the modelled turbulence kinetic energy, and $\delta_{ij}$ the Kronecker symbol. Equations 1 and 2 are valid for any of the models tested in this work. However, as we will describe below, the meaning of $\langle \Phi \rangle$ and $\tau_{ij}(v_i, v_j)$ of the selected models is significantly different.

2.1 Reynolds-Averaged Navier-Stokes Equations

In the RANS equations for statistically unsteady flows, ensemble averaging is applied to the flow variables and to the continuity and momentum equations. This means that all turbulence
fluctuations are modelled and that the dependent variables are mean flow quantities, i.e. statistics to decompose instantaneous quantities into mean and turbulence components have been done \textit{a priori}.

In RANS, \(\tau_{ij}(v_i, v_j)\) are the so-called Reynolds stresses, which are modelled in this work using three different turbulence models: the two-equation \(k-\omega\) SST eddy-viscosity model; \(k-\omega\) SST model combined with local-correlation transition model \(\gamma - \text{Re}_\theta\) and the explicit algebraic Reynolds stress model (EARSM) based on the two-equation \(k-\omega\) TNT eddy-viscosity model. It must be acknowledged that none of these models was developed to simulate statistically unsteady flows. Therefore, there is no guarantee that the turbulent diffusion term obtained from these models is sufficient to suppress all turbulent fluctuations so that the computed velocity field corresponds to the mean flow.

2.1.1 \(k-\omega\) SST Model

The \(k-\omega\) SST model \cite{10} is an isotropic eddy-viscosity model widely used in hydrodynamic applications that solves two transport equations to calculate the turbulence kinetic energy, \(k\), and specific dissipation, \(\omega\). The eddy-viscosity is calculated from

\[
\nu_t = \frac{a_1 k}{\max\{a_1 \omega; \langle S \rangle F_2\}},
\]

where \(\langle S \rangle\) stands for the mean flow strain-rate magnitude, \(a_1\) is a constant and \(F_2\) an auxiliary function that is given with the remaining constant and functions of the model in \cite{10}.

2.1.2 \(\gamma - \text{Re}_\theta\) Transition Model

It is well known that the \(k-\omega\) SST model does not predict transition at the correct location, see for example \cite{3}. The \(\gamma - \text{Re}_\theta\) transition model (or Local-Correlation Transition Model, LCTM) \cite{9} is combined with the \(k-\omega\) SST model to improve the prediction of transition. The definition of \(\nu_t\) remains unchanged (equation 4), but the production and dissipation terms of the \(k\) transport equation depend on the effective intermittency \(\gamma_{eff}\) that is obtained from \(\gamma\) and \(\gamma_{sep}\), which is related to separation-induced transition.

2.1.3 \(k-\omega\) TNT Explicit Algebraic Reynolds-Stress Model

In the EARSM model by Dol et. al \cite{2} a corrective extra anisotropy tensor, \(a_{ij}^{(ex)}\), is added to equation 3,

\[
\frac{\tau_{ij}(v_i, v_j)}{\rho} = 2\nu_t \langle S \rangle - \frac{2}{3} k \delta_{ij} - a_{ij}^{(ex)} k.
\]

The selected EARSM model relies on the \(k-\omega\) TNT model \cite{8} that calculates the eddy-viscosity from \(\nu_t = \frac{k}{\omega}\).

2.2 Delayed Detached-Eddy Simulation

DDES \cite{5} combines RANS in near-wall regions with an SRS model in outer and detached regions. Therefore, the meaning of the dependent variables changes from mean flow in the RANS
region to spaced filtered flow quantities in the SRS region\(^3\). As a consequence, statistics must be applied a posteriori to the SRS flow field to obtain the mean flow field, which may not be a trivial exercise if the goal is to perform ensemble averaging.

DDES achieves the change from RANS to an SRS approach by changing the dissipation term of the \(k\) transport equation to reduce \(k\) and consequently \(\nu_t\). This modification of the \(k\) transport equation changes the meaning of \(\tau_{ij}(v_i, v_j)\) from the Reynold stress in the RANS region to the sub-grid scale stress of the SRS region.

A turbulent length scale, \(l_t\),

\[
l_t = l_{RANS} - f_d \max \{ l_{RANS} - l_{LES}; 0 \}, \tag{6}
\]

is introduced in the dissipation term of the \(k\) transport equation, where \(l_{RANS} = \sqrt{k} / (\beta^* \varpi)\) is the RANS length scale and \(l_{LES} = C_{DES} \Delta\) is the LES length scale. \(C_{DES}\) is a constant and \(\Delta\) is the largest cell characteristic length. The blending between the RANS and SRS regions is obtained from the \(f_d\) empiric function \((0 \leq f_d \leq 1)\) defined as

\[
f_d = 1 - \tanh \left( \frac{C_{d_1} \nu_t + \nu}{\kappa^2 d^2 \sqrt{0.5 (\langle S \rangle^2 + \langle \Omega \rangle^2)}} \right)^{C_{d_2}}. \tag{7}
\]

where \(\langle \Omega \rangle\) is the resolved vorticity magnitude, \(\kappa\) is the Von-Kármán constant, \(d\) is the wall distance and \(C_{d_1}\) and \(C_{d_2}\) are constants given in [5].

### 2.3 Extra Large-Eddy Simulation

XLES [7] uses a similar approach, combining the RANS \(k - \omega\) TNT model [8] with an LES \(k\) sub-grid scale model. To this end, the turbulent length-scale is defined as

\[
l_t = \min \left\{ \frac{\sqrt{k}}{\omega}; C_1 \Delta \right\}, \tag{8}
\]

and the eddy-viscosity by \(\nu_t = l_t \sqrt{k}\). In equation 8, \(C_1\) is a constant [7].

### 2.4 Partially-Averaged Navier-Stokes Equations

PANS uses the same mathematical model for the complete flow field. It relies on a RANS turbulence model and two parameters that define the percentage of the turbulence quantities that is modelled, \(f_\phi = \phi/\Phi\) (\(f_\phi = 1\) corresponds to RANS and \(f_\phi = 0\) to DNS). Determination of mean flow quantities requires the application of statistics \textit{a posteriori}, which is only straightforward to do for time-averaged quantities. In this study we selected the PANS formulation of the \(k - \omega\) SST model [17], that uses constant values for \(f_k\) and \(f_\varepsilon\) \(f_\omega = f_\varepsilon / f_k\). Unlike the previous SRS models, this model has no direct dependency on the spatial or temporal resolution, permitting to separate numerical and modelling errors.

\(^3\)The discussion of the effect of ignoring commutation errors is out of the scope of this paper.
3 TEST-CASES AND NUMERICAL SETUP

3.1 Case 1: Circular Cylinder at \( Re = 3,900 \)

The flow around a circular cylinder at \( Re = 3,900 \) is a common benchmark case for turbulence modelling owing to the low Reynolds number and availability of experimental data. At this \( Re \), a laminar boundary-layer detaches from the cylinder’s surface originating a free shear-layer. Laminar-turbulent transition occurs in this shear-layer and the wake is fully turbulent. The reference data used in this work is taken from [13–15]. The computational domain mimics the dimensions of the experimental apparatus of [15], except for the span-wise dimension. It is a rectangular prism defined in a Cartesian coordinate system centred at the cylinder axis. The inlet of the computational domain is located 10\( D \) upstream of the cylinder, whereas the outlet is 40\( D \) downstream. The cross-section of the domain is \( 24D \times 3D \) (height (\( L_2 \))× wide (\( L_3 \))). The inlet turbulence intensity, \( I \), is 0.2%.

3.2 Case 2: Circular Cylinder at \( Re = 140,000 \)

The main features of this flow are similar to those of the previous case at \( Re = 3,900 \). However, as the \( Re \) increases, laminar-turbulent transition moves upstream. Although at \( Re = 140,000 \) transition still occurs in the free shear-layer, it takes place in the vicinity of the cylinder. The experimental studies of [1,14] are used for comparison. The computational domain used in this case is equal to that used for the \( Re = 3,900 \) case. The exception is the span-wise length, \( L_3/D \), that is reduced to 2.0. The inlet turbulence intensity is equal to that reported in [1], \( I = 0.1\% \).

3.3 Cases 3 and 4: Rounded Square Prism at \( Re = 100,000 \) and 0°/45° of incidence

These cases are part of the CFD workshop on Code Validation of High Reynolds Number Flow around a Square Column with Rounded Corners [6]. The selected cases are the flow around a rounded square prism at \( Re = 100,000 \) and incidence angles of 0 and 45 degrees. The radius, \( r \), of the rounded corners is 16% of the square length/diameter, \( D \). The experimental measurements were carried out on in a cryogenic wind tunnel with a cross squared section 10\( D \) wide. The numerical simulations use a computational domain centred on the prism’s axis with a cross-section of \( 10D \times 3D \) (\( L_2 \times L_3 \)). The inlet and outlet boundaries are located 8.3\( D \) upstream and 88.3\( D \) downstream the prism’s axis. Turbulence intensity is set equal to the experimental value of \( I = 0.3\% \).

3.4 Numerical Settings

All cases addressed employed multi-block structured grids, figure 1, with spatial resolutions ranging from \( 3.0 \times 10^6 \) to \( 22.4 \times 10^6 \) cells. The selected time-steps varied from \( 6.0 \times 10^{-3} \) to \( 1.5 \times 10^{-3} \) time-units, which guaranteed an instantaneous maximum Courant number smaller than 2.0. The details of the spatial-temporal resolutions chosen for this paper are summarized in table 1.

All numerical simulations started from solutions of a precursor 200 time-units RANS simu-
lation. Then, the calculations ran for a period between 200 and 500 time-units\(^4\). Depending on the case, the first 50 to 100 time-units were discarded. The simulations were carried out in double precision and use an iterative convergence criteria, \(c_{it}\), that requires a maximum normalized residual\(^5\) of \(10^{-5}\) for all transport equations at each time-step. The discretization of the governing equations relied on both temporal and spatial second-order accurate schemes, including the convective terms of all transport equations. Naturally, the mathematical models varied with the test-case. Cases 1 and 2 employed RANS using the SST model with or without the LCTM model (case 2) and the EARSM model (case 1), the hybrid models DDES and XLES.

\(^{4}\)These simulation times are larger than what is normally seen in the literature and have been considered in order to minimize the statistical uncertainty of the results.

\(^{5}\)Normalized residuals are equivalent to dimensionless variables changes in a simple Jacobi iteration.
and the bridging model PANS solving 75% of the turbulent kinetic energy, $K$, field ($f_k = 0.25$ and $f_\epsilon = 1.00$). On the other hand, case 3 was simulated with RANS supplemented by SST and EARSM and PANS resolving 50% of $K$ ($f_k = 0.50$ and $f_\epsilon = 1.00$), whereas case 4 relied on RANS using SST and EARSM and DDES. The reasoning for the selection of different models for each case is explained later. All results shown below are normalized\textsuperscript{b} by the inflow stream-wise velocity, $V_\infty$, fluid density, $\rho$ and cylinder/prism diameter, $D$.

<table>
<thead>
<tr>
<th>Case</th>
<th>$L_1/D$</th>
<th>$L_2/D$</th>
<th>$L_3/D$</th>
<th>$N_c$</th>
<th>$N_3$</th>
<th>$\Delta t V_\infty/D$</th>
<th>$\Delta TV_\infty/D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>50.0</td>
<td>24.0</td>
<td>3.0</td>
<td>3,011,736</td>
<td>42</td>
<td>$5.976 \times 10^{-3}$</td>
<td>500</td>
</tr>
<tr>
<td>2</td>
<td>50.0</td>
<td>24.0</td>
<td>2.0</td>
<td>22,367,520</td>
<td>240</td>
<td>$1.500 \times 10^{-3}$</td>
<td>200-500</td>
</tr>
<tr>
<td>3</td>
<td>96.7</td>
<td>10.0</td>
<td>3.0</td>
<td>9,384,000</td>
<td>120</td>
<td>$2.500 \times 10^{-3}$</td>
<td>200-500</td>
</tr>
<tr>
<td>4</td>
<td>96.7</td>
<td>10.0</td>
<td>3.0</td>
<td>9,384,000</td>
<td>120</td>
<td>$2.500 \times 10^{-3}$</td>
<td>200-500</td>
</tr>
</tbody>
</table>

Table 1: Numerical settings of the simulations.

3.5 CFD Code

ReFRESCO (www.refresco.org) is a community based open-usage CFD code for maritime problems. It solves multiphase incompressible viscous-flows using the Navier-Stokes equations (filtered), complemented with turbulence models, cavitation models and volume-fraction transport equations for different phases. The equations are discretized using a finite-volume approach with cell-centred collocated variables, in strong-conservation form, and a pressure-correction equation based on the SIMPLE algorithm is used to ensure mass conservation. Time integration is performed implicitly with first or second-order backward schemes [25]. The implementation is face-based, which permits grids with elements consisting of an arbitrary number of faces and if needed $h$-refined grids. For turbulent flows, RANS and SRS approaches can be used. State-of-the-art CFD features such as moving, sliding and deforming grids, as well automatic grid adaptation (refinement and/or coarsening) are also available. The code is parallelized using MPI and subdomain decomposition, and runs on Linux workstations and HPC clusters. ReFRESCO is currently being developed, verified and applications validated at MARIN (the Netherlands) in collaboration with Instituto Superior Técnico (Portugal), Texas A&M University, and several other universities around the world.

4. RESULTS

4.1 Case 1

The flow around a circular cylinder at $Re = 3,900$ has been addressed in multiple studies as demonstrated in the literature survey presented in [17, 21]. The low $Re$, simple geometry, and availability of experimental data are the reasons for such interest. The results obtained for the time-averaged drag coefficient, $\overline{C_D}$, base pressure coefficient, $\overline{C_{pb}}$, recirculation length

\textsuperscript{b}Reference kinematic viscosity is equivalent to $(V_\infty D)/Re$. 

429
The data demonstrates the limitations of traditional isotropic RANS models to predict such class of flows. Except for the $St$ which is reasonably well predicted by all formulations, the SST model leads to large comparison errors, $E_c(\phi) = \phi - \phi_{\text{exp}}$. The minimum $E_c(\phi)$ occurs for $C_D$ and is larger than 27% of the experimental value. On the other hand, the employment of the EARSM substantially improves the quality of the predictions. The application of SRS models, in turn, further improves the simulations, with the XLES and PANS models attaining an excellent agreement with the experiments.

Integral quantities can be however influenced by modelling errors cancelling and so the assessment of the modelling accuracy based only on these quantities is often biased. Therefore, local quantities have also been analyzed. Time-averaged stream-wise $\langle V_1 \rangle$ and transverse velocity $\langle V_2 \rangle$ fields in the near-wake region, as well as pressure distribution on the cylinder’s surface, $C_p(\theta)$, are shown in figure 2. The results show once more that the SRS approaches, and in this case all of them, present a better agreement with the experimental data than the RANS approaches.

Figure 3 emphasizes the differences between RANS and SRS for the time-averaged cylinder wake field: different circulation zone length, wake axial-velocity deficit and external flow acceleration close to the body which mirrors the different pressure distributions presented in figure 2. Figure 4 presents iso-surfaces of normalized $Q$ (also known as $Q$-criterion), defined as $Q = 1/2 \left( \Omega^2 / S^2 - 1 \right)$ with $\Omega$ being the vorticity rate and $S = \langle S \rangle$ the strain rate, for the same instant with respect to lift time history. Results obtained with RANS exhibit three-dimensional effects in the mean flow, which would be obviously missed with a 2-D assumption. Naturally, XLES exhibits totally different flow structures than RANS in the wake due to the instantaneous character of its dependent variables.

### 4.2 Case 2

The flow around a circular cylinder at $Re = 140,000$ is characterized by transition in the free shear-layer in the vicinity of the cylinder. Considering that the majority of turbulence models was formulated assuming a fully turbulent state and predict transition too early in terms of Reynolds number [3], this case is expected to be challenging. This affects not only the RANS models but also some SRS approaches due to the fact that they do not solve turbulence in the locations where transition occurs (e.g. near-wall regions).

---

$^7$It must be pointed that $C_L'$ may have different meanings in the experiments and in the selected mathematical models.
The predicted $C_D$, $C_{pb}$, $C'_L$, and $St$ are shown in table 3. RANS with SST leads to a poor prediction of the aforementioned quantities. The magnitude of $E_c(\phi)$ is larger than 30% for all the selected quantities. This misrepresentation of the flow is caused by an early transition prediction that occurs in the boundary-layer instead of in the free shear-layer. To further illustrate this point, table 3 has experimental measurements taken at $Re = 5.0 \times 10^6$. It is visible that the RANS/SST predictions match reasonably well with such data. Naturally, this issue is also present in SRS models that do not resolve turbulence in the boundary-layer. Therefore, the
results of DDES and XLES show values of $E_c(\phi)$ similar to RANS/SST. On the other hand, the use of LCTM reduces the differences between numerical predictions and experimental data for the selected flow quantities. Furthermore, by resolving a fraction of the turbulent spectrum in the boundary-layer, PANS is able to significantly improve the results. This is also demonstrated in figure 5a for the pressure distribution on cylinder’s surface. Whereas PANS achieves low comparison errors, the remaining formulations lead to large values of $E_c(C_p(\theta))$.

Figure 5b depicts the stream-wise velocity, $\langle V_1 \rangle$, predicted with RANS/SST, LCTM, and PANS. It shows that despite the quality of PANS ($f_k = 0.25$) to predict to quantities given in table 3 and $C_p(\theta)$, the time-averaged profile of $V_1$ at $x_2/D = 0$ is poorly captured. This deserves further investigation.

Finally, figure 6 presents instantaneous $Q$ iso-surfaces for RANS and PANS solutions. It shows that RANS captures three-dimensional vortical structures in the shear-layer, as for the previous test case. On the other hand, this figure also illustrates the capabilities of a SRS bridging approach like PANS, once the spatial and temporal resolution is fine enough to model the relevant physics of the flow; PANS not only resolves a very large spectrum of turbulence scales in the shear-layer but also in the boundary-layer. This advocates the future use of PANS even for the simulation of boundary-layer transitional flows.
<table>
<thead>
<tr>
<th>Model</th>
<th>$C_D$</th>
<th>$C_{pb}$</th>
<th>$C'_L$</th>
<th>$St$</th>
</tr>
</thead>
<tbody>
<tr>
<td>SST</td>
<td>0.72</td>
<td>0.83</td>
<td>0.35</td>
<td>0.244</td>
</tr>
<tr>
<td>LCTM</td>
<td>1.08</td>
<td>1.24</td>
<td>0.62</td>
<td>0.203</td>
</tr>
<tr>
<td>DDES</td>
<td>0.65</td>
<td>0.77</td>
<td>0.03</td>
<td>0.264</td>
</tr>
<tr>
<td>XLES</td>
<td>0.68</td>
<td>0.75</td>
<td>0.05</td>
<td>0.264</td>
</tr>
<tr>
<td>PANS</td>
<td>1.26</td>
<td>1.32</td>
<td>0.52</td>
<td>0.188</td>
</tr>
<tr>
<td>Exp.</td>
<td>1.24</td>
<td>1.21</td>
<td>0.52</td>
<td>0.179</td>
</tr>
<tr>
<td>Exp.²</td>
<td>0.74</td>
<td>0.93</td>
<td>-</td>
<td>0.257</td>
</tr>
</tbody>
</table>

Table 3: Time-averaged drag coefficient, $C_D$, base pressure coefficient, $C_{pb}$, recirculation length, $L_r$, root-mean-square lift coefficient, $C'_L$, and Strouhal number, $St$, as a function of mathematical model. Case 2. Exp.² indicates experiments at $Re \approx 5 \times 10^6$ [23].

Figure 5: Time-averaged stream-wise velocity, $\langle V_1 \rangle$, at $x_2/D = 0.0$, and pressure distribution on the cylinder’s surface, $C_p(\theta)$, as a function of the mathematical model. Results for case 2.

Figure 6: Instantaneous $Q$ iso-surfaces ($Q = 0.1, 0.2$ and $0.5$) in the near wake using the SST and PANS models. Results for case 2.

4.3 Cases 3 and 4

The last cases addressed in this work are the flows around a rounded square prism at angles of incidence of 0 and 45 degrees. The quantitative results are shown in table 4. The results obtained with RANS using EARSM and DDES/PANS demonstrate a significant reduction of $E_c(\phi)$ when compared to those of the SST model. It is also interesting to observe the similarity
of results between RANS/EARSM and PANS or DDES models. Although experimental results are not available for the time-averaged stream-wise velocity field, $\langle V_1 \rangle$, in the near-wake, and pressure distribution on the prism’s surface, $C_p(\theta)$, the data shown in figure 7 presents similar trends between models.

![Table 4](image-url)

Table 4: Time-averaged drag coefficient, $C_D$, pressure coefficient at two different locations, $C_{p1}, C_{p2}$, recirculation length, $L_r$, and Strouhal number, $St$, as a function of mathematical model. Cases 3 and 4.

It is important to mention the reason why different SRS models are used in cases 3 and 4. The use of a hybrid formulation is advantageous to attain a good trade-off between modelling accuracy and numerical resources, especially for high Reynolds numbers. However, the application of this type of formulation should be avoided if the boundary/shear-layer cannot be modelled accurately by RANS, or in case a critical region of the flow is located in the transition between the RANS and SRS modes. The former issue might occur in case 3 owing to the proximity of the crucial shear-layer to the prism’s wall. Consequently, the authors preferred to use a SRS formulation able to resolve turbulence in the near-wall region (PANS) in this case. For case 4, due to the large detached free shear-layer hybrid methods should be adequate.

Figure 8 presents $Q$ iso-surfaces for the RANS/SST and DDES solutions for case 4, zoomed closer to the structure than for the previous cases. The results show that for this 45 degree orientation RANS/SST predicts a shear layer with more three-dimensionality and less vortical coherence than for the previous test cases, especially in the near-wake. Since the boundary-layer detaches from the structure into a large free shear-layer, the DDES model activates its LES character in these layers, permitting to solve turbulence, even if these are close to the body.

### 4.4 General Remarks

From the previous results and experience on the selected test cases, some general remarks can be made:

- While for cases 1 and 2 the numerical errors seem to be under control, and the spatial and temporal resolutions used adequate for the problems at hand (verification studies have been performed but not shown here) for cases 3 and 4, both grids, time-steps and computational times need further refinement.

- When shear-layers are important, large and even if close to bodies, hybrid SRS approaches are a viable choice to increase the fidelity of the calculations when compared with linear...
Figure 7: Time-averaged stream-wise velocity, $\langle V_1 \rangle$, and pressure distribution on the prism’s surface, $C_p(\theta)$, as a function of the mathematical model. Results for cases 3 and 4.

Figure 8: Instantaneous $Q$ iso-surfaces ($Q = 0.1, 0.2$ and $0.5$) in the near wake using the SST and DDES models. Results for case 4.

RANS approaches. When laminar-turbulent transition occurs in these layers, and the spatial and temporal resolutions are fine enough, transition might be captured by hybrid SRS methods.
• When laminar-turbulent transition is predicted in the near-wall region (e.g. inside the boundary-layer), hybrid SRS methods will act as RANS and so they suffer from the same problems as the underlying RANS turbulence models. Bridging methods, that solve turbulence also inside the boundary-layer are theoretically more adequate to tackle these problems, at an increasing computational cost. For fully turbulent unsteady boundary layers around blunt bodies, the current linear RANS turbulence models derived for statistically steady flows are not accurate enough. In this situation, both hybrid and bridging models should be applied.

• Ideally, SRS methods should be engineering-safe methods where for coarse temporal and spatial resolutions they should fall back to their RANS formulations. However, for mildly separated/unstable flows DDES/XLES methods may suffer from the well-known “grey-area” problem [11]. For PANS, the results are highly dependent on the filter size $f_k$, and a wrong combination of grid, time-step and filter may lead to large modelling errors; these cannot be chosen independently of each other, even if numerical and modelling errors are not entangled. This means that a three-dimensional convergence study (filter, grid size and time-step size) is needed to fully assess modelling accuracy, such as done in [21].

• EARSM, being a non-linear RANS model delivered improved accuracy for all cases tested here. The LCTM transition model also improves the modelling of boundary-layer and shear-layer transition when compared with the RANS solution using the underlying turbulence model.

• In terms of computational costs, for the same grid and time-step, table 5 shows the number of iterations per time step, needed to reach the desired maximum normal residual $10^{-5}$ for all quantities solved. This number, in general, increases from RANS towards PANS. Additionally, for EARSM the anisotropy tensor has to be extra calculated and has a strong influence on the iterative convergence of the momentum equations, and LCTM (not shown in Table 5) solves two more transport equations. Therefore, all approaches tested are more computationally expensive than RANS using the SST model. Note also that in the context of the segregated SIMPLE method employed, the pressure equation needs also more iterations to achieve the convergence tolerance when using EARSM, LCTM or SRS approaches.

• Table 6 shows rough estimates of computational times for case 1 and case 2. The results are estimated for MARIN Marclus4 2014 HPC cluster (each computational node has 2 Xeon(R) CPU E5-2660-v3-2.60GHz 10-cores CPUs, and an HBA Infiniband for inter-nodal communication). The results indicate that, for the same total simulation time, RANS-EARSM calculations are more expensive than RANS-SST but less than SRS, and that PANS-SRS calculations are more expensive than DDES ones. The RANS computational times here shown are obviously exaggerated, since in this case usually 150 time-units are enough to have statistic convergence in time-averaged quantities. However, for SRS calculations, simulation times larger than 300 time-units were imperative for the present test cases. For EARSM, shorter simulation times are usually needed than for SRS, but more research is needed before general guidelines can be made.
Table 5: Averaged number of iterations per time step as a function of case and mathematical model. * denotes $f_k = 0.50$ instead of $f_k = 0.25$.

Table 6: Estimated computational costs as a function of case and mathematical model. Computational time per time step, $T_{\Delta t}$, total simulation time, $T_{sim} = \Delta TV_\infty / D$ (see Table 1), number of processes, $N_p$, wall-clock time, $Cpu$, and total core-days computational time, $CoreDays$.

Estimates for MARIN Marclus4 2014 HPC cluster.

5 CONCLUSIONS

This study investigates the modelling accuracy of RANS and SRS models to simulate shear-layer flows around blunt bodies. The selected formulations are the isotropic RANS $k - \omega$ SST with or without the LCTM, an anisotropic RANS EARSM, DDES, XLES and PANS. These models were assessed on four cases: the flow past a circular cylinder at $Re$ of 3,900 and 140,000, and the flow around a rounded square prism at $Re = 100,000$ and angles of incidence of 0 and 45 degrees. The outcome of the present study leads to the following conclusions:

- The modelling accuracy of traditional isotropic RANS models is insufficient to simulate the class of flows addressed in this study.

- The EARSM model used in this investigation leads to a substantial improvement of the quality of RANS simulations. For boundary-layer transition-dominated flows, the usage of LCTM transition models also improves the RANS results.

- In general, the SRS models tested in this work show a good agreement with the experiments, specially for the two first cylinder cases. For the rounded square prism case more research is needed.

- The application of hybrid formulations such as DDES or XLES is not justified if boundary-layer transition phenomena are critical.

- The numerical demands of RANS increase significantly from the SST to the EARSM, and from the RANS/EARSM to the SRS tested, if one considers not only the computational costs per time-step but the total number of time-steps needed (total simulation time) to reach statistically converged results.
Further work is needed in order to finalize the complete assessment of these models for the rounded square prism case. Currently, all turbulence approaches presented here are also being applied for flows around foils, 3D wings, cavitating propeller flows, offshore constructions, free-surface flows and manoeuvering ships. Verification and validation studies of all these cases will help to derive guidelines for the usage of these new turbulent approaches for maritime industry relevant flows.

ACKNOWLEDGMENTS

This research is partially funded by the Dutch Ministry of Economic Affairs. This support is gratefully acknowledged.

REFERENCES

STUDY OF UNSTEADY HYDRODYNAMIC EFFECTS IN THE STERN AREA OF RIVER CRUISERS IN SHALLOW WATER

IVAN SHEVCHUK* and NIKOLAI KORNEV

*Chair of Modeling and Simulation, University of Rostock
Albert-Einstein Str. 2, 18059, Rostock, Germany
ivan.shevchuk@uni-rostock.de

Key words: Shallow water, hybrid RANS/LES, unsteady propeller loadings, vibration

Abstract. The main objective of the present paper is the assessment of the influence of shallow water conditions on the hydrodynamic exciters of ship hull vibration using hybrid RANS/LES methods. The following topics will be briefly discussed:

- Validation of hybrid methods for calculation of ship wake in shallow water
- Analysis of the influence of different factors on the nominal wake characteristics
- Assessment of thrust fluctuations and pressure pulses and their sensitivity to motion conditions

It will be shown, that the depth restriction may influence the periodic forces and moments acting on the ship stern and by these means intensify the vibration.

1 INTRODUCTION

One the most important factors for design of river cruise ships is the passenger comfort. However, experience shows, that ship motion in fairways with depth restriction often leads to a rise of strong vibration at the stern which impairs the habitability conditions. This forces ship designers to either avoid placement of passenger cabins in the stern area, or to use additional damping construction to suppress the vibration. Both measures result in increase of costs and reduction of ship economic efficiency. The development of technical measures for vibration reduction requires understanding of main physical mechanisms leading to amplification of parasitic oscillations.

The main reason of hydrodynamically excited vibration is the propeller, which in multiple ways affects the hull. First, the forces and moments, produced by the propeller may considerably fluctuate in time because of the wake nonuniformity and instationarity. Second, the propeller
blades rotating in water create pressure pulses on the hull and by these means cause the oscillations of the hull surface. The pressure pulses due to cavitation usually play the major role for vibration excitation [9]. If cavitation is not present, the lift part is dominant.

In this work the modelling of cavitation and the structural response to the described unsteady effects will left out of the scope in order to keep the computational effort on the feasible level. Thus present work deals only with the hydrodynamic excitation forces, not including the cavitation effects. As an object for the study the generic geometry of the inland cruise vessel is considered.

URANS approach, widely used for industrial applications nowadays, can provide satisfactory results for many tasks like resistance prediction, ship motion in waves, open water propeller tests, etc. [5, 2]. However, when it comes to the unsteady flows, URANS tends to smear the flow features in time and space [3]. This fact motivated the application of hybrid RANS/LES methods for the considered task.

2 VALIDATION

Before applying the computational methodology to the case of interest it was necessary to compare the performance of different hybrid methods for the similar conditions. For this purpose the PIV measurements of the wake of an inland ship (model scale 1:40) in shallow water, provided by the Duisburg Development Centre for Ship Technology and Transport Systems(DST) were used [6]. The considered conditions are summarized in Table 1, where $B_c$ is the breadth of the towing tank, $U$ - model speed, $h$ - water depth, $T$ - ship draught, $F_{n_{h}} = \frac{U}{\sqrt{gh}}$ - depth Froude number. The stern geometry can be seen in Figure 1.

![Stern geometry of M1926 model (left half). The plane, where the velocity field was measured in, is shown at $x = 0.48$.](image)

![Figure 1: Stern geometry of M1926 model (left half). The plane, where the velocity field was measured in, is shown at $x = 0.48$.](image)

Model dimensions were $L_{oa} = 11m$, $B = 1.14m$, $T = 0.25m$. No propeller or any other appendages except the shaft were present in experiment. The velocity was measured in the propeller plane.

The nominal wake $w = 1 - |\langle u_x \rangle|/U$ was calculated from the ensemble-averaged measured values of the longitudinal velocity and compared to the computed ones using contour plots (see Figure 5b, c and d) and one-dimensional plots along the circle $r/R = 0.7$ in the propeller plane (see Figures 2a, b and c).

<table>
<thead>
<tr>
<th>Par-r</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$B_c$ [m]</td>
<td>9.8</td>
</tr>
<tr>
<td>$U$ [m/s]</td>
<td>1.24</td>
</tr>
<tr>
<td>$h$ [m]</td>
<td>0.5</td>
</tr>
<tr>
<td>$T$ [m]</td>
<td>0.25</td>
</tr>
<tr>
<td>$h/T$ [-]</td>
<td>2</td>
</tr>
<tr>
<td>$F_{n}$ [-]</td>
<td>0.12</td>
</tr>
<tr>
<td>$F_{n_{h}}$ [-]</td>
<td>0.56</td>
</tr>
<tr>
<td>$Re$ [-]</td>
<td>$1.36 \cdot 10^7$</td>
</tr>
</tbody>
</table>

Table 1: Parameters of the model and the experimental conditions considered in the validation study.
Following turbulence models were used for validation: RANS $k - \omega$ SST, and SST-IDDES [4]. The description of the governing equations of the SST-IDDES model can be found in the corresponding papers and is omitted here. The averaged in time values of the wake were compared between different models for gradually refined hexa-dominant meshes (3M - 20M cells). For hybrid simulations the timestep was chosen from the condition $Co < 1$ (for each cell), which is an essential constraint for scale-resolving simulations.

![Figure 2](image)

**Figure 2:** Comparison of the wake at $r/R = 0.7$ obtained on different grids using different methods to experiment: (a) - SST-IDDES for different mesh resolutions, (b) - SST-RANS on the same meshes, (c) - comparison of SST-RANS and SST-IDDES.

An important aspect of application of scale-resolving methods is the generation of the turbulent content. In the present work this was done using the modal turbulence generator, proposed in [1]. The turbulence generator in form of the momentum source was placed at a distance of $2(h - T)$ upstream from the end of the parallel middle body at the stern (see Figure 3). The statistics for the generator were taken from the preliminary RANS computation.

The SST-IDDES model and the synthetic turbulence generator were implemented by authors in OpenFOAM CFD toolkit, version 2.3.x and used in the solver *pimpleFoam*.

![Figure 3](image)

**Figure 3:** Vortical structures ($\lambda_2 = -20$) in the stern of M1926 predicted using SST-IDDES + synthetic turbulence generator. One can see, where the turbulent structures are added to the flow.

![Figure 4](image)

**Figure 4:** Dependence of the standard deviation of the longitudinal velocity $\sigma_u$ in percent of the inflow velocity for different mesh resolutions at the points, lying on in propeller plane at $z=-0.23$.

From the plots, presented in Figures 2 and 5 one can see, that RANS $k - \omega$ SST model as...
well as SST-IDDES model predictions are in a good agreement with each other and with the experimental data. SST-RANS model turns out to give decent results for the considered flow. One can explain this by the absence of strong separations in the wake. However, as can be seen in Figure 5, the RANS model smears out the anchor-like form of the wake. The second problem, which one can mention - is the absence of the peak of the wake fraction at $\phi \approx 270^\circ$. The RANS solution seems to be insensitive to the mesh refinement.

The predictions, delivered by SST-IDDES model for the wake are of nearly the same quality as those, computed using SST-RANS. Moreover, in some regions the hybrid model is advantageous: it prevents the smearing of the wake form and is obviously able to capture the peak of the wake fraction at $\phi \approx 270^\circ$. Unfortunately, this is true only for the coarsest mesh. Mesh refinement in this case caused slight deterioration of the results in the region $225^\circ < \phi < 300^\circ$, which is a surprising result. It indicates either the malfunctioning of the model shielding in response to mesh refinement or influence of mesh quality on the results (unstructured mesh was used).

![Figure 5](image.jpg)

**Figure 5:** Comparison of the wake obtained on the different grids using different methods to experiment: (a) - experiment, (b) - SST-RANS on the fine mesh (c) - SST-IDDES on the coarse mesh, (d) - SST-IDDES on the fine mesh

Nevertheless, the predictions of the hybrid model are in close agreement with RANS for the mean velocity field, which shows, that no negative effects, related to modeled stress depletion [8] are present. At the same time the solution, obtained from the hybrid model is fully unsteady and contains the coherent vortical structures (see Figure 3). This allows for evaluation of the effect of the unsteadiness of the wake on the propeller, which will be conducted in the following
sections. From this point of view it is interesting to see, how the mesh resolution influences the standard deviation of the longitudinal velocity field, \( \sigma_u = \frac{1}{|u_\infty|} \sqrt{\frac{1}{n-1} \sum_{i=0}^{n-1} (u_{x,i} - \langle u_x \rangle)^2} \times 100\% \).

As can be seen in Figure 7, \( \sigma_u \) is quite sensitive to the mesh resolution in some points \((y=0, y=0.08)\), however, in the most of the considered points the variations between the meshes are less than 1%.

From the conducted validation study in was concluded, that the implementation of the SST-IDDES model works well for the considered flow: it provides almost the same time-averaged wake fraction as RANS, which are close to experimental data, while containing the unsteady vortical structures.

### 3 ANALYSIS OF THE WAKE OF A RIVER CRUISER

#### 3.1 Nominal wake

First step of the application of the previously validated computational method was the analysis of the nominal wake of a river cruiser at full scale. The ship \((L_{wl} = 135\text{m}, B = 12\text{m}, T = 2\text{m})\) was considered moving in a shallow channel with \(h/T = 1.25, 1.5, 2.0\) at a speed of 3 m/s with the drift angles \(\beta = 0^\circ, 10^\circ\).

Total number of cells used in computations was 20-22M depending on the depth. Stern region was filled with uniform isotropic mesh with the cell sizes 0.04\(\times\)0.04\(\times\)0.04 m\(^3\) in order to improve the performance of the hybrid method in the region of interest. As in the validation studies, the turbulence generator was applied upstream from the stern.

The ship has two azimuth thrusters installed on each side (see Figure 6). A thruster near the skeg will be in the following referred to as the inner (first) and the one near the bilge - as outer (or second). In the nominal wake computations the analysis concentrated on the evaluation of the wake nonuniformity and instationarity, which can contribute to the unsteady loadings on the propellers.

From the Figure 8 one can draw a number of important conclusions. Firstly, the wake in the disc of the outer thruster is much lower in all cases and is almost insensitive to the fairway

---

**Figure 6:** Vortical structures revealed in the wake of the river cruiser using \(\lambda_2\) criterion. Motion with the drift angle \(\beta = 10^\circ\), \(h/T = 1.5\). With propellers.

**Figure 7:** Vortical structures revealed in the wake of the river cruiser using \(\lambda_2\) criterion. Motion with the drift angle \(\beta = 10^\circ\), \(h/T = 1.5\). Without propellers.
I. SHEVCHUK and N. KORNEV

Figure 8: Plots of the resolved Reynolds stresses $\langle u'_x^2 \rangle$ (left) and wake (right).

depth. For the inner one this is not the case - the depth reduction leads to the rapid increase of the wake. Secondly, when the depth is reduced the velocity fluctuations tend to be concentrated near the inner thruster, whereas the outer one works in an almost undisturbed velocity field. No strong separations were observed at the stern at straight course.

The flow picture changes considerably, when the motion with the drift angle $\beta = 10^\circ$ is considered. In this case the skeg vortex becomes very strong at smaller depths. The level of velocity fluctuations at the inner thruster increases, the amplitude of fluctuations reaches up to 15-20% of the inflow value. The wake is on the first pod is decreased, but at the second one - considerably grows and in addition the irregularity of the velocity field is much stronger.

3.2 Working thrusters

After the analysis of the nominal wake, simulations with the rotating propellers were conducted for the same conditions, as listed above. Only the propellers form the port side were geometrically resolved, whereas the ones on the starboard side were replaced by the actuator
disks, producing the prescribed thrust (see Figure 6). As the analysis showed, the wake and the resolved turbulence kinetic energy are very small on the starboard side and are very unlikely to cause any unsteady effects on the thrust.

Based on the propeller forces history, obtained during the simulations, the statistical analysis of thrust fluctuations was performed. The values of the maximum amplitude of fluctuations and their standard deviation $\sigma_T$ are shown in Tables 2 and 3.

Table 2: Statistics of the total thrust oscillations on the inner (1) and outer (2) pods for the motion at straight course

<table>
<thead>
<tr>
<th>$h/T$</th>
<th>1.25</th>
<th>1.5</th>
<th>2.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pod</td>
<td>1</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>$T'_{\text{max}} / \langle T \rangle \times 100$ [%]</td>
<td>5.78</td>
<td>5.07</td>
<td>5.25</td>
</tr>
<tr>
<td>$\sigma_T \times 100$ [%]</td>
<td>2.44</td>
<td>2.07</td>
<td>2.26</td>
</tr>
</tbody>
</table>

Table 3: Statistics of the total thrust oscillations on the inner (1) and outer (2) pods for the motion at $\beta = 10^\circ$

<table>
<thead>
<tr>
<th>$h/T$</th>
<th>1.25</th>
<th>1.5</th>
<th>2.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pod</td>
<td>1</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>$T'_{\text{max}} / \langle T \rangle \times 100$ [%]</td>
<td>4.05</td>
<td>5.15</td>
<td>3.33</td>
</tr>
<tr>
<td>$\sigma_T \times 100$ [%]</td>
<td>1.34</td>
<td>2.08</td>
<td>0.90</td>
</tr>
</tbody>
</table>

In the tables one can observe the following trends. For the straight course the maximum fluctuation amplitude and the standard deviation of the thrust increase on the first pod as the depth gets smaller. On the outer pod reverse trend is observed. This means, that when the depth is decreased, the thrust fluctuations tend to be more intense of the inner pod. Motion with the drift angle surprisingly led to reduction of both statistical parameters on the first and on the second pod. This can be explained by the fact, that when the ship is moving with the drift angle, the turbulent structures, generated on the hull follow the flow direction and therefore are not transported directly into the propeller disc.

Analysis of the pressure fluctuations on the hull above the propellers revealed, that their amplitude is increasing from 3kPa to 7kPa (for $\beta = 0^\circ$) and to 8kPa (for $\beta = 10^\circ$) as the $h/T$ ratio decreases from 2.0 to 1.25. This effect is pronounced for both propellers equally.

Even though the present research is concerned mostly with the unsteady thrust oscillations, the fact, that the cavitation may play a major role for the pressure oscillations on the hull (and the vibration) is acknowledged. Therefore, in order to evaluate, whether the cavitation can occur, an a-posteriori analysis of the cavitation inception regions was conducted. The regions, on the propeller blades, where the pressure is smaller than the vapour pressure $p_v$ are shown in Figure 9 for different motion conditions. There are two most important trends, which can be noticed. Firstly, the size of the regions $p < p_v$ decreases as the $h/T$ ratio gets smaller. However, in this case these regions are equally large on all blades. Secondly, when the motion with $\beta = 10^\circ$
is considered, a significant difference in the size of low pressure zones on different blades can be observed. This effect stems from the fact, that the propeller works in an oblique flow, where the effective angle of attack depends on the blade position. Dramatic change of cavitation inception regions depending on the blade position allows to conclude that collapsing of cavitation sheets is likely to occur. This scenario of vibration intensification is described in [7].

![Figure 9: Cavitation inception regions on the front propeller of the inner thruster for different motion conditions.](image)

(a): $h/T = 1.25, \beta = 0^\circ$  
(b): $h/T = 1.5, \beta = 0^\circ$  
(c): $h/T = 1.25, \beta = 10^\circ$

4 CONCLUSIONS

From the presented computational results, one can conclude, that the unsteady effects at the stern of cruise vessels are intensified, as the depth-to-draft ratio decreases. For the nominal wake, strong velocity oscillations and wake irregularities were observed at small depths. Simulations with the rotating propellers revealed the growth of the maximum fluctuation amplitude and the standard deviation of thrust in response to depth reduction (for inner pod at straight course and for both pods at $\beta = 10^\circ$). The amplitude of the pressure pulses on the hull increases from 3kPa at $h/T = 2.0$ to 7-8kPa at $h/T = 1.25$. Finally, it was shown that the cavitation may occur, which would contribute to the pressure pulses on the hull plating. All of these factors can contribute to vibration intensification and therefore have to be accounted for during the design process of inland cruise vessels.

5 ACKNOWLEDGEMENTS

The support of the Mecklenburg State Graduate Foundation and the German Research Foundation (Deutsche Forschungsgemeinschaft) under the grant INST 264/113-1 FUGG is gratefully acknowledged. The authors would like to thank Mr. Benjamin Friedhoff and Prof. Bettar el Moctar from the Duisburg Development Centre for Ship Technology and Transport Systems (DST) for access to validation data and many helpful discussions.

REFERENCES


AN IMU AND USBL-AIDED BUOY FOR UNDERWATER LOCALIZATION

B. Allotta*, M. Bianchi*, F. Fanelli*, J. Gelli*, N. Monni*, M. Pagliai*, N. Palma* and A. Ridolfi*

*Department of Industrial Engineering (DIEF), University of Florence
Via di Santa Marta 3, 50139, Florence, Italy
e-mails: a.ridolfi@unifi.it, francesco.fanelli@unifi.it, m.pagliai@unifi.it

Key words: Underwater Robotics, Underwater Localization

Abstract. Autonomous underwater navigation remains, as of today, a challenging task. The marine environment limits the number of sensors available for precise localization, hence Autonomous Underwater Vehicles (AUVs) usually rely on inertial and velocity sensors to obtain an estimate of their position either through dead reckoning or by means of more sophisticated navigation filters (such as Kalman filters and its extensions [1]). On the other hand, acoustic localization makes possible the determination of a reliable vehicles pose estimate exploiting suitable acoustic modems [3]; such estimate can even be integrated within the navigation filter of the vehicle in order to increase its accuracy. In this paper, the authors discuss the development and the performance of an Ultra-Short BaseLine (USBL)-aided buoy to improve the localization of underwater vehicles. At first, the components and the physical realization of the buoy will be discussed; then, the procedure to compute the position of the target will be analyzed. The following part of the paper will be focused on the development of a recursive state estimation algorithm to process the measurements computed by the buoy; specifically, Extended Kalman Filter [4] has been adopted to deal with the nonlinearities of the sensors housed on the buoy. A validation of the measurement filtering through experimental tests is also proposed.

1 INTRODUCTION

Undersea operations are an example of how robotics can replace humans: working underwater is, indeed, both dangerous and difficult. The birth of the underwater robotics is due to military purposes (e.g. seabed mines clearing), but with years the field of application has widened to a vast category of scientific and commercial tasks. An example is the digital reconstruction of the seabed, exploiting a proper set of sensors mounted on board an underwater vehicle, like waterproof cameras and multibeam echosounders. Due to the heterogeneity of tasks that can be performed, many types of underwater vehicles exist: among them it is possible to find Remotely Operated Vehicles (ROVs), i.e. tethered underwater mobile devices operated by a crew aboard a vessel and Autonomous Underwater Vehicles (AUVs), capable of travelling underwater without requiring input from an operator. By using such vehicles the need of a proper localization system
arises [5]. In fact, radio waves, exploited by Global Positioning System (GPS) are quickly absorbed by the water, hence the need to rely on different instruments for the positioning. Such instruments are usually based on acoustic waves because of their property to propagate in the water even for long distances; in this sense, the localization techniques mainly used in the underwater environment are Long BaseLine (LBL) and Ultra Short BaseLine (USBL). In Long BaseLine localization, different transmitter modems are placed underwater in well known positions. Periodically, pings are generated by these modems, as response to acoustic signals coming from the target to locate (i.e. the underwater vehicle), and are received back by the modem mounted on the vehicle. Once responses from all the transmitters are available, they are used to retrieve the position of the vehicle, by triangulation or position search algorithms. LBL can reach an accuracy of a few centimetres, a precision that does not depend on the distance between the vehicle and the fixed transmitters. On the other hand, LBL systems require a not negligible amount of work for the installation of the baseline stations; this procedure, indeed, often requires vessels and proper equipments.

The Ultra Short BaseLine localization, instead, is based on a single device integrating both the acoustic transceiver (needed for communicating with the compatible acoustic modems) and a series of transducers (typically 5), placed in known relative positions (normally in the order of few tens of centimetres). Transducers are capable of acquiring the signal transmitted by a modem and, once the signal reaches them, it is acquired and processed by a dedicated unit. A typical installation for the USBL device is on the bottom of a surface vessel [5], with a considerable saving on the costs of the infrastructures that have to be installed in the operational area compared to the LBL solution. However, the positioning accuracy guaranteed by such a system degrades with the relative distance and additional sensors (e.g. GPS, gyroscopes or electronic compass) are required, in order to compensate for the changing position and orientation of the surface vessel the device is mounted on. Despite these drawbacks, USBL solutions are widely employed for tasks where flexibility and time saving in the installation are mandatory.

In this paper underwater target localization is discussed, exploiting a self-moving buoy, built by the Mechatronics and Dynamic Modelling Laboratory (MDM Lab) of the University of Florence, housing an USBL device. Section 2 provides the hardware features of the buoy and a description of the procedure used for locating the target. In Section 3 the development of an Extended Kalman Filter (EKF) to improve the measurements computed by the buoy is discussed. The solution proposed has been validated through data obtained from experimental tests on stationary and moving targets; the results are presented in Section 4.

2 DESCRIPTION OF THE BUOY

2.1 Hardware features

The buoy used for underwater localization is shown in Figure 1: it consists of a wooden board fixed on a life buoy and a case located on top. The case is certified with the IP67 (International Protection) rating, meaning that the inside is impermeable on condition that the immersion into the water is temporary, not deeper than 1 meter and not longer than 30 minutes. Cases with higher IP ratings can be found in commerce, but the IP67 rating is suitable for this purpose, being the case located over the water level. Almost all of the water-sensitive (electronic) components of the buoy are placed inside the IP67-rated case and connected to the
Figure 1: The buoy used to locate the underwater target (Figure 1a) and the IMU-USBL subsystem (Figure 1b).

outside by using impermeable cables and cable glands suitable for aquatic environment. Two thrusters can also be installed in order for the buoy to reach or maintain a desired position. Regarding the electronic and electrical features, the following components are installed on the buoy:

- Six cells, 8000 mAh (Ampere-hour) Lithium-ion Polimer (Li-Po) battery;
- 24V-5V DC-DC converter;
- Odroid-XU onboard computer;
- Xsens MTi-100 Inertial Measurement Unit (IMU);
- EvoLogics S2CR 18/34 USBL transceiver;
- SwiftNav Piksi Differential GPS (DGPS) receiver;
- PicoStation Ubiquiti wireless module;
- Polulu Micro Maestro 6-channel USB servo controller.

To guarantee the impermeability of the unit, the Xsens IMU is located inside a watertight aluminum cylinder which is rigidly fixed on the USBL transceiver; the USBL transceiver is mounted on a stainless steel pole rigidly connected to the wooden board of the buoy. Both IMU and USBL transceiver are positioned underwater. Thanks to the rigid connection between the IMU-USBL system and the buoy, every variation in the orientation of this latter is reflected in a same variation in the one of the IMU-USBL system and it will be detected by the IMU; this
information is then used to properly process the measurement provided by the USBL sensor. The Piksi DGPS receiver communicates with a base station, located in a known position on the mainland and provides the position of the buoy with an accuracy of few centimetres.

Regarding the electronic components, the 8000 mAh Li-Po battery provides 24V voltage to supply the onboard computer and, consequently, some of the components that are plugged to it. A 24V-5V DC-DC converter has been inserted in order to avoid overvoltages for the Odroid-XU computer. Both the wireless module and the DGPS receiver are fixed on a tinier wooden board on top of the main one and connected to it by means of two 50 cm threaded rods. This solution has been adopted in order to limit interferences in the radio signals sent and received by the buoy. The Polulu Micro Maestro 6-channel USB servo controller provides an USB interface for the two thrusters.

In order to power on the onboard computer of the buoy, a magnetic activation switch is used; this solution guarantees easy emergency shutdowns when buoy components safety may be at risk.

2.2 Localization procedure

The procedure to compute the position of an underwater vehicle involves an Earth-fixed frame, located in a given position in the sea and whose axes are aligned according to the NED (North East Down) convention; in this context, the Earth-fixed frame origin is coincident with the first position of the buoy measured by the DGPS.

In order to detect variations in the pose (i.e. position and orientation) of the buoy with respect to the Earth-fixed frame, the IMU and the DGPS are used. Let us consider a buoy-fixed reference frame; of course, at first such frame will be coincident with the Earth-fixed frame, but it will change its pose according to the movements of the buoy. The DGPS measures the position of the buoy-fixed frame origin \(O_B\) with respect to another reference frame, fixed on the base station located on the mainland; let us indicate such measurement with \(O^B_S\). By calling
the position of the origin of the Earth-fixed frame (measured by the DGPS and referred to the base station) it is trivial to compute the relative positioning of the buoy with respect to the Earth-fixed frame \( \mathbf{P}_B^W \) as:

\[
\mathbf{P}_B^W = \mathbf{O}_B^s - \mathbf{O}_W^w.
\]  

(1)

Regarding the orientation, the IMU measures the relative orientation between the buoy-fixed and the Earth-fixed frame, expressed with the RPY (Roll Pitch Yaw) Euler angles \( \varphi, \vartheta, \psi \). Therefore, given the position of the target measured by the USBL \( \mathbf{P}_U^T \) and known the relative orientation of the USBL sensor with respect to the IMU, expressed by the mounting angles \( \varphi_m, \vartheta_m, \psi_m \), it is possible to compute the position of the target with respect to the current position of the buoy \( \mathbf{P}_B^T \) as:

\[
\mathbf{P}_T^B = \mathbf{R}_B^b \mathbf{R}_U^I \mathbf{P}_U^T.
\]  

(2)

where \( \mathbf{R}_B^b \) and \( \mathbf{R}_U^I \) are, respectively, the rotation matrices describing the relative orientation between the IMU and the buoy and between the IMU and the USBL device. The expression of \( \mathbf{R}_B^b \) can be computed starting from the RPY angles \( \varphi, \vartheta, \psi \) by using the rotation matrices composition rule; the same holds for \( \mathbf{R}_U^I \) and the mounting angles \( \varphi_m, \vartheta_m, \psi_m \).

The position of the target referred to the Earth-fixed frame can then be calculated as:

\[
\mathbf{P}_T^W = \mathbf{P}_T^B + \mathbf{P}_B^W.
\]  

(3)

Known \( \mathbf{P}_T^W \) and the absolute position (latitude, longitude and altitude) of the Earth-fixed frame origin \( \mathbf{O}_W \), the absolute position of the target can be computed using standard conversion functions as follows:

\[
\mathbf{P}_T^{LLA} = f_c(\mathbf{O}_W, \mathbf{P}_T^W).
\]  

(4)

where \( f_c(\cdot, \cdot) \) is the function performing the conversion from relative to absolute coordinates.

To compute the position of the vehicle, measurements obtained from the sensors mounted on the buoy (i.e. IMU, USBL and DGPS) are processed using Robot Operating System (ROS) [7].

Specifically, for each sensor a ROS node implements the interface by publishing data on a proper ROS topic: the content of such topics is then read by another node that computes the position of the underwater target as discussed.

3 Measurements Filtering

To improve the underwater localization, a recursive state estimation algorithm can be used; such an algorithm relies on a model of the target to locate and of the sensors exploited to compute its position. Given the nonlinearities of the sensors involved, the Extended Kalman Filter (EKF) [4] has been used.

3.1 Underwater target motion model

Two different motion models need to be considered: one describing a stationary target and another referred to a target moving at constant speed. These are, indeed, the two main situations...
occurring in the practice: usually an AUV navigates at the cruise speed along its minimal resistance direction or is required to maintain a fixed position (i.e. hovering).

The mathematical description exploits the relative coordinates of the target referred to the Earth-fixed frame, but considering only the North and East components; the depth (Down axis component) is not relevant for this topic as it is usually known or can be measured properly by the depth sensor housed on board the vehicle.

For these reasons, the considered state vector is $x = [x_N, y_N]^T$ for the stationary target and $x = [x_N, \dot{x}_N, y_N, \dot{y}_N]^T$ for the target travelling at constant speed.

### 3.1.1 Stationary target state space representation

The state space representation describing the behaviour of a stationary target can be obtained directly at discrete time. Ideally, the equation modelling the evolution of a quantity $x$ that does not change over time is:

$$x_{t+1} = x_t.$$  \hspace{1cm} (5)

However, in order the model to be more realistic, a white noise $w_t$, having minimal standard deviation $\sigma$, is added:

$$x_{t+1} = x_t + w_t , \quad w_t \sim wn(0, \sigma^2 I).$$  \hspace{1cm} (6)

### 3.1.2 Moving target state space representation

To model the behaviour of a target moving at constant speed polynomial kinematic models described in [6] are adopted. Specifically, a White Noise Acceleration (WNA) model is used. Such a model is derived starting from a continuous time motion model and then discretized using the ZOH (Zero Order Hold) technique for the discretization.

Let us consider the components of the target motion $x = [x_N, \dot{x}_N, y_N, \dot{y}_N]^T = [x^1, x^2, x^3, x^4]^T$; assuming different acceleration fluctuations ($\sigma_{cx}$ and $\sigma_{cy}$) along the two directions ($x^1$ and $x^3$), it is possible to model a target moving at constant speed with the following equations:

$$\begin{bmatrix}
  x^1_{k+1} \\
  x^2_{k+1} \\
  x^3_{k+1} \\
  x^4_{k+1}
\end{bmatrix}
= \begin{bmatrix}
  A_k & 0_{2 \times 2} \\
  0_{2 \times 2} & A_k
\end{bmatrix}
\begin{bmatrix}
  x^1_k \\
  x^2_k \\
  x^3_k \\
  x^4_k
\end{bmatrix}
+ w_k ,$$  \hspace{1cm} (7)

$$w_k \sim wn(0, Q_k), \quad Q_k = \begin{bmatrix}
  \sigma_{cx}^2 Q_{k} & 0_{2 \times 2} \\
  0_{2 \times 2} & \sigma_{cy}^2 Q_{k}
\end{bmatrix}$$

where:
with $T_k = t_{k+1} - t_k$ being the offset between the $k$-th and $k+1$-th time samples. It is important to note that $T_k$ varies over time, because the working frequency of the USBL sensor is not fixed. This is caused by the time required from the acoustic signal to travel from the USBL transceiver to the acoustic modem and back to the USBL transceiver, which varies depending on the water conditions and the distance between the two devices.

### 3.2 Measurement equations

As it has been discussed earlier, the underwater localization procedure is based on data provided by IMU, USBL and DGPS. However, the angles measured by the IMU are already compensated by its inner estimation algorithm. The DGPS, instead, measures the position of the buoy; if the absolute position of the base station is reliable, the DGPS measurements are accurate. On the other hand, the USBL transceiver provides a position of the underwater target that can be inaccurate for many reasons (e.g. buoy position perturbations or bad water conditions): the filtering is then aimed to the compensation of the USBL measurements errors.

Let us consider the case of the stationary target (for the moving one the procedure is similar); let us indicate with $x_{W}^B$, $y_{W}^B$ the North and the East coordinate of the buoy at time $t$ with respect to the Earth-fixed frame. Considering that the USBL transceiver exploits a spherical positioning system, the characteristic of the sensor set used to locate the target can be modelled as:

$$
y_t = h_t(x_t) + v_t = \begin{bmatrix} \sqrt{(x_1^t - x_{W}^B)^2 + (x_2^t - y_{W}^B)^2} \\
\text{atan2}(x_1^t - x_{W}^B, x_2^t - y_{W}^B) \end{bmatrix} + v_t \tag{9}
$$

where $v_t$ is a zero-mean Gaussian noise depending on the USBL transceiver technical features.

The $\text{atan2}(\cdot, \cdot)$ function in Equation (9) provides an information about the azimuth angle of the target and is piecewise continuous and differentiable; when differentiable, its partial derivatives are:

$$
\frac{\partial \text{atan2}(x, y)}{\partial x} = -\frac{y}{x^2 + y^2}, \quad \frac{\partial \text{atan2}(x, y)}{\partial y} = \frac{x}{x^2 + y^2}. \tag{10}
$$

The points where $\text{atan2}(x, y)$ is discontinuous ($x = 0 \land y \neq 0$) or undefined ($x = 0 \land y = 0$) never occur in practice, hence it is always possible to compute its partial derivatives; the range measurement characteristic $h_t^1(x_t)$, instead, is always differentiable. It is then possible to use the Extended Kalman Filter to process the measurements provided by the buoy.

Let us note that the choice to adopt the Earth-fixed frame as reference frame leads to a time-variant measurement equation: the coordinates of the buoy $x_{W}^B$ and $y_{W}^B$, indeed, vary over time. However, this is not an issue, because the expression of the partial derivatives of $h_t(x_t)$ remains the same, therefore it can be computed only once, offline, and applied online, reducing the computational load required from the EKF.

The expression of the Jacobian matrix $C_t$ to be used within the filter is then:
Figure 3: Satellite image of the piers of Roffia Lake (from Google Maps): the piers are labelled as ‘P1’, ‘P2’, ‘P3’, ‘P4’, and ‘P5’. In the performed tests the buoy has been located at the end of P1 and two acoustic modems are moored at different piers (P2 and P5).

\[
C_t = \begin{bmatrix}
\frac{e_x}{\sqrt{(e_x^2 + e_y^2)}} & \frac{e_y}{\sqrt{(e_x^2 + e_y^2)}} \\
-\frac{e_y}{e_x^2 + e_y^2} & \frac{e_x}{e_x^2 + e_y^2}
\end{bmatrix}
\]  

(11)

where:

\[
e_x = (\hat{x}_1^{t|t-1} - x^W_B), \quad e_y = (\hat{x}_2^{t|t-1} - y^W_B)
\]

(12)

being \(\hat{x}_t|t-1 = [\hat{x}_1^{t|t-1}, \hat{x}_2^{t|t-1}]^T\) the predicted state estimate at time \(t\).

4 EXPERIMENTAL RESULTS

To evaluate the effects of the measurements filtering on the underwater target localization several experimental tests were carried out at Roffia Lake (San Miniato, PI). Stationary and moving target localization have been performed exploiting, respectively, two EvoLogics 18/34 acoustic modems moored in known positions and MARTA AUV (MARine Robotic Tool for Archaeology [2], Figure 5a). The measurements have been collected online, while the filtering has been applied offline, using MATLAB, in order to find a proper tuning of the EKF parameters.

4.1 Stationary target localization

In the stationary target localization (see Figure 3), two EvoLogics 18/34 acoustic modems have been moored at the piers P2 and P5 of Roffia Lake at a given depth, while the self-moving buoy has been placed near pier P1.

In the experiments in exam various angular perturbations have been induced on the buoy, so that it has been possible to verify the robustness of both the raw measurements and the filtered ones. Figures 4a and 4b show the result of the localization of the targets moored at P2 and P5, respectively. The measurements computed by the buoy have been compared with the position of the target measured on surface with a GPS. It is possible to note how the localization of the target results in a wide circular sector; this is due to the high yaw angular rates caused by the perturbations voluntarily induced on the buoy. By applying the filtering of the USBL measures, the localization performance is improved. In fact, as visible in Figure 4d, the azimuth angle errors obtained after the filtering are considerably concentrated with respect to those of the raw
measurements. It is important to note that the errors mean is not zero: however, this is given by the inaccuracy of the target GPS measure (usually around 3 meters) used as benchmark.

4.2 Moving target localization

The localization of a moving target exploiting the buoy has been tested using MARTA AUV. Specifically, different paths have been performed by the vehicle, in order to fully evaluate the performance of the localization filter: in the following the results obtained executing a lawn-mower path (Figure 5b) are discussed.

For every line composing the path, a steady speed is reached after an initial transient: the lawn-mower path is then, ideally, composed of subsequent uniform linear motions along different lines. On the other hand, the moving target model is referred to an uniform linear motion along a single line, hence a major importance has been given to the EKF update step rather than the prediction one through a suitable tuning of the EKF parameters, in order to trust more the measurements with respect to the target model.

The target positions and velocities estimated by the filter have then been evaluated. Specifically, the estimated speeds have been compared with the measurements of the Doppler Velocity Log (DVL) sensor housed on board the vehicle. As for the estimated positions, in this context it has not been possible to use the position of the target provided by the GPS as benchmark, being such sensor unavailable underwater. The estimated positions, then, have been evaluated in terms of smoothness, because the path described by the vehicle is usually regular; indeed, the water strongly dampens the vehicle motion.

In Figure 6a it is possible to note that the filtered positions are more regular than the raw ones, hence more consistent with the trajectory followed by the target. As for the velocities, the comparison with data obtained from DVL is shown in Figure 6b. It can be noted that the velocities estimated by the EKF present a slight time delay with respect to those measured on board the vehicle. The authors believe that this behaviour is caused by the time needed by the acoustic waves to propagate underwater, which produces a time offset between the position computed by the buoy at a given time and the actual position of the vehicle. Also, an initial transient for the speed estimate is present, depending on the initial state estimate the filter is initialized with. Despite these drawbacks, the results obtained through the use of the moving target model within the EKF are satisfying.

5 CONCLUSIONS

This paper focuses on underwater target localization exploiting an USBL transmitter housed on a self-moving buoy, aided with an IMU and a DGPS. Firstly, the hardware features of the buoy have been introduced; then, the procedure used by the buoy to compute the position of the underwater target has been presented. To improve the quality of the measurements computed by the buoy, a recursive state estimation algorithm has been applied: in particular, Extended Kalman Filter has been used to deal with the nonlinearity of the USBL characteristic. Regarding the state transition equation of the filter, two different models have been used. The first one results sufficiently accurate to describe the behaviour of a stationary target, while the second one models a moving target and has been obtained by using a White Noise Acceleration model, suitable for the case of an uniform linear motion. Such a strategy has been validated through
Figure 4: Localization of a stationary target. Figure 4a refers to the modem located at P2, Figure 4b to the one located at P5. It can be noted that the localization results in a wide circular sector, due to the perturbations applied to the buoy. Figure 4c highlights how the use of a state estimation filter improves the localization of the target located at P2 with respect to the raw measurements; such a result is remarked in Figure 4d, reporting the distribution of the errors on the azimuth angle.
Figure 5: MARTA AUV (Figure 5a) and the onboard estimate of the lawn-mower path followed (Figure 5b).

Figure 6: Figure 6a shows a detail of the estimate of the North coordinate in the lawn-mower path: as it is possible to notice, the measurements filtering improves the smoothness of the positions computed. Figure 6b reports a comparison between the estimated velocities and those measured on board the vehicle by the DVL.
experimental tests, conducted exploiting two EvoLogics 18/34 acoustic modems and MARTA AUV. It has been seen that, by filtering the measurements, the localization performance is improved, both in terms of stationary targets (reduction of the azimuth error angle dispersion) and moving targets (improved smoothness of the estimated positions). In the latter case, the velocities estimated by the EKF have been compared with the measures provided by the DVL sensor on board MARTA AUV. Such a comparison highlighted that the estimates of the AUV speeds are consistent with the DVL measurements, but affected by a slight time offset, given by the time required from the acoustic waves to propagate underwater. The results are, however, promising.

Possible future developments may concern the implementation of a multiple model filtering, based on both the moving and the stationary target models, in order to allow the outcome of the localization filter to be more robust, especially when complex paths are performed by an underwater vehicle.

6 ACKNOWLEDGEMENTS

This work has been supported by the European project BRUCE, that has received funding from the European Union’s Seventh Framework Programme within “SUNRISE - Open Call 2 for new beneficiaries”, under grant agreement no 611449 [8].

REFERENCES


FIRE PERFORMANCES OF A MARINE BULKHEAD: NUMERICAL EVALUATION OF THERMOMECHANICAL BEHAVIOR

VIRGINIE DREAN*, GUILLAUME CUEFF* AND GILDAS AUGUIN*

*Efectis France
19-23 quai de Paludate, 33800 Bordeaux, France
e-mail: virginie.drean@efectis.com, www.efectis.com

Key words: Numerical simulation, Marine product, Fire performances, CFD, FEM

Abstract. The fire performances of marine products are assessed with large scale fire resistance tests following dedicated Standards. However, regarding the conditions of such normative tests, orientation studies for research and development purposes are limited. The aim of the present study is now to develop a numerical model to investigate others configurations. Using the fire performance results of a given bulkhead achieved by fire resistance tests, extrapolation of thicknesses, material properties, joint configuration, etc. can be numerically assessed and used to validate or orientate the final configuration to be tested.

1 INTRODUCTION

The fire performances of marine products are assessed with large scale fire resistance tests following dedicated Standards [1]. Considering marine bulkheads, fire resistance furnaces are used. The performed tests allow the measurements of temperature criteria on the unexposed side of bulkhead, and the evaluation of the panel deflection. However, regarding the conditions of such normative tests, orientation studies for research and development purposes are limited using the results of such a test.

The aim of the present study is now to develop and validate a numerical model in order to study several configurations of the tested product. Using the fire performance results of a given bulkhead achieved by fire resistance tests, extrapolation of thicknesses, material properties, joint configuration, etc. are assessed numerically and used to validate or orientate the final configuration to be tested.

In a previous project, the numerical model of a virtual fire resistance furnace designed with the CFD code FDS [3] has been validated for partition walls [4] and wooden doors [15]. A fairly good agreement was found for each quantity to validate the hypothesis of the developed numerical model [5]. In the same manner, the virtual facility is adapted for marine application. The predicted thermal loads are applied as boundary conditions on the exposed side of a marine bulkhead modelled with the FEM code SAFIR [6]. The considered bulkhead is constituted 4 panels made with 2 steel sheets and an inner layer of mineral wool, with a total thickness of 25 mm, and joint with continuous steel plates. The thermal properties of each constitutive material have been implemented.

The numerical results achieved for the thermomechanical behaviour of the marine bulkhead are compared with experimental ones. The global agreement allows further extensions studies of the product (dimensions, materials, design...). The developed numerical tool is then validated for such application and a strong coupling between FEM and CFD codes will be addressed.
3 DESCRIPTION OF THE STUDIED MARINE PRODUCT

In this study, the marine product considered is a bulkhead provided by the manufacturer MAPAC Panel [7].

The fire performances of this bulkhead were evaluated during the test named 14V025 [8] following the Standards [1] requirements. A second fire resistance test named 14V030 [9] was ordered by the manufacturer as an orientation test for research improvements.

3.1 Description of the specimen

The bulkhead is realized by 4 panels “25T ECO” type. The individual panel dimensions are 600 x 2472 x 25 mm (width x height x thickness). Each panel is made out by a layer of mineral wool with density of 168 kg/m³ and a 23.6 mm thickness, inserted and glued between two steel sheets with a 6/10 mm thickness on the unexposed side and a 45/100 mm on the exposed side. On the edges, each steel sheet realized a 8.5 mm folding.

Figure 1: Fire test of a marine bulkhead – classification test 14V025 [8] and orientation test 14V030 [9]

For the test 14V025 [8], the junctions between panels are realized with a continuous steel plate with a 20/10 mm thickness folded in “H” shape with overall dimensions of 34 x 15 mm and inserted into a groove created into the edges of the panels. A gap of 2 mm exists between the panels on the unexposed side.

Concerning the orientation test 14V030 [9], the "H" studs have been replaced flat steel sheets. The central junction between the panels is realized by a continuous steel plate with a 20/10 mm thickness and a 50 mm length inserted into a groove created into the edges of the panels. Laterally, the panels are equipped with a closing steel profile with a 10/10 mm thickness and folded in “U” shape with a section of 25 x 26 x 25 mm. The bulkhead is tested with fire opposite to the vertical “H” profile during test 14V025 as indicated in the Figure 1.
3.2 Description of the frame

The bulkhead is blocked inside a concrete frame aperture by means of 8 steel angles (4 on each face) with a 20/10 mm thickness and folded in “L” shape with a section of 50 x 50 mm and 150 mm high fixed to the concrete by dowels after interposition of mineral wool between the closing profiles and the concrete frame. On the upper and lower parts, the panels are fixed together by means of a steel profile folded in “U” shape with 10/10 mm thickness. The supporting frame has dimensions of 2440 x 2500 x 200 mm (width x height x thickness).

2 EVALUATION OF THE FIRE PERFORMANCES OF A MARINE BULKHEAD

This section is dedicated to the implementation of a fire resistance test for marine bulkhead. The fire resistance furnace and the dedicated instrumentation are described.

2.1 Fire resistance furnace

The fire performances of marine products such as bulkhead are assessed with large scale tests performed in fire resistance furnaces with different designs by accredited laboratories and following dedicated Standards [1].

These tests must comply with requirements of European Standard EN 1363-1 [2], which impose conventional values of relative static pressures and temperatures at 100 mm from the exposed side of the tested elements. Two constrains must be achieved simultaneously:
- The static overpressure must be maintained to 20 Pa at the top of the vertical tested element, like bulkhead;
- The thermal program inside the furnace is defined by a time dependant logarithmic curve ranging from 20°C at the start of the test to approximately 1050 °C after 2 hours of test (see equation (1), with T in °C and t in minutes).

\[ T = 345 \log_{10}(8t + 1) + 20 \]  

In the furnace, the instrumentation consists in six plate thermometers placed at 100 mm of the exposed side of the tested specimen to control the thermal elevation indicated in Figure 2. These plate thermometers are constituted by an Inconel steel sheet insulated on its backside by a refractory board. An Inconel thermocouple is welded on the Inconel steel sheet. The pressure inside the furnace is controlled continuously using a probe located at the head of the vertical tested specimen. During the test, the temperature in the laboratory is also recorded.

![Figure 2: Fire resistance test of a marine bulkhead – thermal elevation in the furnace [2]](image-url)
2.2 Fire performances of a marine bulkhead

The tested bulkhead is installed as the closure wall of the large industrial fire resistance furnace and a specific instrumentation is implemented on its unexposed side as indicated on the Figure 3. The instrumentation consists in thermocouples placed on the unexposed side of the specimen, at intersection and quarters of the diagonal of the panel, and at 15 mm of the junction of the panels. The bulkhead deflection is measured by potentiometric sensors.

The temperature criteria on the unexposed side of bulkhead allow the evaluation of the fire performances of the specimen. They are based on a maximal temperature rise of 225°C and/or an average temperature rise of 140°C on the unexposed side of bulkhead.

Figure 3: Fire resistance test of a marine bulkhead – a) Fire resistance furnace b) Instrumentation of the unexposed side of the bulkhead installed on the furnace [8]

3.3 Description of the instrumentation

For each test, the instrumentation consists in thermocouples placed on the unexposed side of the specimen, at intersection and quarters of the diagonal of the panel, and at 15 mm of the junction of the panels. The bulkhead deflection is measured by potentiometric sensors. Furthermore, during the orientation test 14V030 [9], additional displacement and temperature sensors have been added, particularly in the thickness of the bulkhead. The instrumentation location for each test is indicated in Figure 4.

Figure 4: Instrumentation location for tests 14V025 [8] and 14V030 [9]
4 NUMERICAL EVALUATION OF THE FIRE PERFORMANCES OF THE MARINE BULKHEAD

Regarding the conditions of the normative fire resistance tests, orientation studies for research and development purposes are limited. In order to study several extrapolations of a primary tested specimen, Efectis is leading a R&D project called VIRGILE which consists in a virtual test furnace simulator.

In this section, the virtual fire resistance test simulator is used to evaluate the fire behavior of the marine bulkhead described in paragraph 3. The numerical method implemented is discussed and validated based on the experimental results acquired during tests 14V025 and 14V030.

4.1 Virtual fire resistance facility

Lot of experimental works has been led on the measurement methods [10][11][12], but, few studies have been presented concerning the simulation of furnaces used for fire resistance tests [13][14].

The VIRGILE project is dedicated to the development of a numerical model for a virtual fire resistance furnace designed by the way of a modified version of the CFD code FDS.5 [3].

Among the different fire resistance facilities simulated in the frame of this project, the modelled so-called V furnace was based on one vertical furnace of Efectis France laboratory. The dimensions of the main chamber of this furnace are 3.1 x 1.5 x 3.6 m (width x length x height). A mesh size of 10 cm³ was employed. A total of 45344 cells are needed to model the furnace, the chimney and the burners. The model of the considered geometry and internal dimensions of the furnace are plotted in Figure 5.

The thermal and physical properties of the furnace constituent materials are taken into account because the heat transfer in the solid walls is computed with FDS 5.

![Outline of the virtual V furnace structure](image)

Figure 5: Outline of the virtual V furnace structure

The chimney flue communicates with the furnace through a rectangular opening on the rear wall of the furnace. At the other end of the pipe, there is the aspiration area, transcribing the effects of the chimney. An opening on the surface of the duct, 1 m behind the flue, is
introduced to set a reference pressure in the computational domain.

The facility is fitted with 12 burners, fed with natural gas. The burner model involves an external combustion cavity. Thus, 12 openings of 0.2 x 0.2 m are installed on lateral sides of the furnace. Air and gas are injected into the external cavities of the furnace, so that the air/methane combustion mixture takes place in these cavities and not in the furnace one. Only the hot smoke from burning is injected into the furnace in the form of jets, as observed on furnaces of Efectis laboratory.

To achieve the thermal program requirements, the adopted approach is based on an iterative method for correcting the outflow of the chimney to regulate the pressure, and the percentage of the burner’s air valve opening to regulate the temperature at each time increment. The CFD code FDS has been modified to introduce an automatic control of the 12 gas burners and another one for the output flow imposed at the chimney exit. The control model was done by considering 3 groups (stages) of 4 burners (two on each side of the furnace). Each floor was considered independent at the air supply and gas (Figure 6).

The control of the air/gas mixing and the hot gases exhaust permits to comply simultaneously with both general conditions imposed by EN 1363-1. The integrated error between the imposed program and the thermal gas temperatures recorded by the Plate Thermometers located 100 mm from the closure wall of the furnace is estimated during the calculation.

The Plate Thermometer model corresponds to that described in detail in the report [11] and used for the virtual V furnace. They are composed of an Inconel stainless steel sheet of thickness of 0.7 mm, and an inorganic insulating refractory plate of thickness of 10 mm. These two plates are nested and form a square of 10 cm length, as shown in Figure 7. The alloy part of each plate thermometer is oriented to the furnace side. This design makes plate thermometers quite sensitive to the radiative heat flux coming from flames of burners and lining of furnaces.
To create a realistic simulator, it has been necessary to model the furnace behavior and its interaction with the tested element by the way of a coupling between this furnace simulator and the element modelled with a FEM code.

In a previous study, a strong coupling between FDS and the FEM code CASTEM [16] has been implemented. An interface has been created between this code and the modified version of FDS 5 to ensure the thermal coupling between the virtual furnace and the element. Thermal constrains delivered by FDS 5 are refreshed regularly on the exposed element surface. These thermal constrains are constituted by a radiative flux coming from the furnace lining, a convective flux due to the hot gases on the vicinity of the element and a radiative flux emitted by the exposed side of the tested element (see Figure 8). Thanks to these constrains the temperatures of the exposed side of the tested element are determined and they constitute the new boundary conditions for the calculation of the thermo-dynamic equilibrium of the furnace inner volume for the next time increment step. This integrated tool has been validated for partition walls [4] and then to study the fire behavior of wooden doors [15]. A fairly good agreement was found for each quantity to validate the hypothesis of the developed numerical model [5].
4.2 Virtual fire resistance test for marine bulkhead

In the same manner, the virtual facility is now adapted for marine application. In this study, a weak coupling between CFD and FEM codes is performed. The thermal loads predicted by the virtual fire resistance furnace modeled with the CFD code FDS are applied as boundary conditions on the exposed side of the marine bulkhead described in paragraph 3 and modelled with the FEM code SAFIR [6]. The thermal properties of each constitutive material have been implemented in the CFD and FEM codes. First, the thermal behaviour of the bulkhead is investigated. The reference tests are reproduced using the V furnace simulator as shown on Figure 9. The correct regulation of the furnace in terms of temperature and pressure control according to the test standard EN 1363-1 is verified (Figure 10).

![Figure 9: CFD simulation of the fire resistance test for the marine bulkhead – a) real test b) virtual test c) temperature field in the burner’s axis](image)

![Figure 10: Furnace regulation for temperature and pressure – experimental values (dotted lines) and numerical results (straight lines)](image)

Then, detailed FEM models of the panels are built to reproduce the material layers as well as the junction elements used in tests 14V025 and 14V030. These models and the corresponding boundary conditions are indicated on Figure 11 in a transverse view of the panels.

The boundary conditions consist in heat fluxes and convective transfer coefficients $H$ evaluated with the virtual fire resistance furnace. These quantities take into account the interaction between the facility and the tested element by the way of the burner regulation. So,
even if the thermal solicitation follows the test standard requirements, the heat fluxes imparted to the tested element depend on its thermal properties. In a same manner, the convective transfer coefficient $H$ will be adjusted depending on environment conditions in the furnace.

![Figure 11: FEM model and boundary conditions for the tested bulkhead with a) “H” steel junction (test 14V025) b) plate steel junction (test 14V030)](image)

To validate the thermal transfer analysis performed, the numerical results obtained in terms of temperature at the unexposed side of the bulkhead are compared with experimental ones for the two reference tests in Figure 12 a). This temperature corresponds to the surface temperature measured in the center of panel 2, far from junctions and impact of hot smoke releases through openings (Thermocouples 19 and 20 in Figure 4). Then, the numerical results obtained in terms of average temperature at the unexposed side of the bulkhead are compared with experimental ones for the two reference tests in Figure 12 b). This average temperature corresponds to the surface temperature of the panels measured at intersection and quarters of the diagonals of the bulkhead (Thermocouples 8 to 12 in Figure 4). Regarding the temperature criterion required, the fire performance of the bulkhead is lost numerically 10 seconds after the experimental time.

![Figure 12: Numerical and experimental temperature at unexposed side of the bulkhead a) far from junctions (center of panels) b) at quarters and intersection of diagonals – fire performance loss criteria](image)
A global agreement is observed and allows further thermomechanical analysis of the bulkhead. For the average temperature evaluated at quarters and intersection of diagonals, the main differences between experimental and numerical values are explained by the hot gas released through openings at junctions. In the numerical model, the junctions are heated faster than the core of the panel because of steel elements and gap between steel and wool layer.

Then, the thermomechanical behaviour of the bulkhead is investigated. The steel profiles modelled are considered blocked in terms of movements in the 3 directions (UX, UY and UZ) at the top and at the bottom. The rotation along the central axis is also locked at both ends to avoid twisting. The mechanical loading applied corresponds to the weight of the profile as well as the section of the panel taken up by this profile, equivalent of a width of 600 mm of panel per profile.

The Figure 13 provides a comparison between numerical deflections evaluated at the half-height displacements of the "H" studs and the steel plate with the experimental deflections measured at the center of the bulkhead during tests 14V025 and 14V030.

The experimental deflections correspond to those of the exposed and unexposed side measured at the center of the panel at half height of the stud. Although test 14V030 is slightly different from the 14V025 test, the order of magnitude for the displacements measured is quite close to the numerical ones. The values at the end of the simulation are respectively 115 mm and 107 mm for the "H" profile and the steel plate, against 139 mm for the experimental measurement at 33 min for the test 14V030 (before the final ruin of the bulkhead). Differences between numerical and experimental results can be explained by different reasons:

- The displacements of the reinforced concrete frame have an impact on the displacement of the central stud of the bulkhead. This impact has not been measured experimentally and is therefore not taken into account.

- During test 14V030 the head of the central steel stud dislodged from its support near the upper rail, allowing a greater displacement towards the fire (phenomenon hardly predicted numerically).
5 CONCLUSIONS

The present study was dedicated to the development of a numerical model to study a product configuration in order to propose further extensions. The numerical model of a virtual fire resistance furnace designed with the CFD code FDS has been validated for marine application. The predicted thermal loads were applied as boundary conditions on the exposed side of a marine bulkhead modelled with the FEM code SAFIR.

The numerical results achieved for the thermomechanical behaviour of the marine bulkhead are compared with experimental ones. The global agreement allows further configurations studies of the product (dimensions, materials, design...). Thus, using the fire performance results of a given bulkhead achieved by fire resistance tests, extensions of thicknesses, material properties, joint configuration, etc. are assessed numerically and used to orientate the final configuration to be tested.

The developed numerical tool is then validated for such application and a strong coupling between FEM and CFD codes will be addressed. By evaluating junction deflection, hot gas release through these openings will be taken into account.

The authors acknowledge MAPAC for its contribution to this research works by providing results from fire resistance tests.

REFERENCES


[16] http://www-cast3m.cea.fr/cast3m/index.jsp, 2010
GEOMETRIC MODEL FOR AUTOMATED MULTI-OBJECTIVE OPTIMIZATION OF FOILS

ELISA BERRINI\textsuperscript{a,b}, BERNARD MOURRAIN\textsuperscript{a}, REGIS DUVIGNEAU\textsuperscript{a}, MATTHIEU SACHER\textsuperscript{c} AND YANN ROUX\textsuperscript{b,d}

\textsuperscript{a}Université Côte d’Azur, INRIA Sophia Antipolis Méditerranée
2004, route des Lucioles - 06902 Sophia Antipolis, France
e-mail: elisa.berrini@inria.fr, bernard.mourrain@inria.fr, regis.duvigneau@inria.fr

\textsuperscript{b} MyCFD - 29 avenue des frères Roustan, 06220 Golfe-Juan, France
e-mail: yann@mycfd.com, elisa@mycfd.com

\textsuperscript{c} Naval Academy Research Institute - IRENAV CC600, 29240 BREST Cedex 9, France
e-mail: matthieu.sacher@ecole-navale.fr

\textsuperscript{d} K-Epsilon - 1300 route des Crêtes, 06560 Valbonne, France
e-mail: yann@k-epsilon.com

Key words: Optimization, design, CAD, foil

Abstract. This paper describes a new generic parametric modeller integrated into an automated optimization loop for shape optimization. The modeller enables the generation of shapes by selecting a set of design parameters that controls a twofold parameterization: geometrical - based on a skeleton approach - and architectural - based on the experience of practitioners - to impact the system performance. The resulting forms are relevant and effective, thanks to a smoothing procedure that ensures the consistency of the shapes produced.

As an application, we propose to perform a multi-objective shape optimization of a AC45 foil. The modeller is linked to the fluid solver AVANTI, coupled with Xfoil, and to the optimization toolbox FAMOSA.

1 INTRODUCTION

Automatic shape optimization is a growing study field, with applications in various industrial sectors. As the performances of a flow-exposed object can be obtained accurately with CFD (Computational Fluid Dynamics), small changes in design can be captured and analysed. To exploit these performance analysis capabilities, it is important to have a precise and efficient control of the geometry of the objects.

To improve the form of a design in order to increase its performances, a precise shape consistency control is essential when performing deformations. Naval architects need to use shape
quality preserving tools to modify hulls or appendages while avoiding non-realistic forms.

The coupling of a flow solver, an optimization algorithm and a quality preserving shape modeller is the basis for an efficient automatic shape optimization loop.

We propose a parametric modeller tool with a new approach for shape deformation, integrated into an automatic shape optimization loop with a flow solver, AVANTI, and an optimization toolbox FAMOSA. The methodology presented here has the ability to generate valid forms from an architectural point of view thanks to an innovative shape consistency control based on architectural parameters. Controlling shapes by architectural parameters allows reducing the number of degrees of freedom of the shape optimization problem. They also introduce a physical and a design meaning into the optimization process.

The approach proposed allows using directly a CAD model based on a NURBS [1] representations in the modeller tool. The methodology developed can be applied to any shape that can be described by a skeleton, e.g. hulls, foils, bulbous bows, but also wind turbines, airships, etc.

This paper presents the general methodology of parametric modeller and an example of application to the shape multi-objective optimization of a AC45 foil, with general form parameters (main lengths, angles, etc.) and local form parameters (foil chord lengths, twist, etc).

2 RELATED WORK

The coupling of a flow solver to a modeller and an optimization algorithm methodology is widely used [2, 3, 4, 5].

Recent technological progress allows running quasi-automatically the CFD solver and post-processing the relevant results of the computation. Optimization algorithms demonstrate their efficiency in solving problems with a large number of degrees of freedom and where the objective function values are difficult and costly to evaluate. However, less efforts have been dedicated to the development of efficient parametric modellers.

Shape deformation for ships or appendages is a relatively recent approach. Classical deformation techniques such as Free Form Deformation (FFD) and morphing, created for 3D animations purposes, have been applied to ship shape optimization [6, 7] (FFD) and [8] (morphing).

Morphing is limited to known bounds of shape variations, the exploration of possible optimal shapes is reduced to a given number of shapes.

FFD can be very efficient if used with a small number of degrees of freedom to control the whole shape of the object. However, local perturbations can be performed only with refinement of the areas of interest, therefore increasing the number of degrees of freedom. FFD does not take into account any architectural parameters when deforming an object, leading possibly to non-realistic results.

Engineering dedicated CAD software provides parametric design features, allowing the user to build parametrized models such as Catia™, Grasshopper for Rhinoceros 3D™ or CAESES from Friendship System™. This method allows generating shapes easily, but all of the parame-
ners are lost when saving the model in a standard format such as IGES or STEP. This represents a limitation for automatic linking with solvers (CFD, structural analysis, etc.) or optimization algorithms.

Specific software have been developed during the last decades for ship applications. One of the most widespread is CAESES\textsuperscript{TM} form Friendship Systems\textsuperscript{TM}, allowing the user to modify imported geometries using advanced geometrical parameters [9]. Similarly, a ship dedicated tool Bataos [10] allows to modify the shape of sections of the hull described B-Spline curves with predefined functions.

These tools are based on geometrical control of shapes. Architectural parameters are computed on the deformed geometry and can be included as constraints, but they do not directly control the shape modification.

3 PARAMETRIC MODELLER

To obtain smoothly deformed shapes, we propose a novel modeller tool based on a generic methodology to modify shapes with architectural constraints. To achieve this objective, we use a twofold parametrization of the shape that allows describing a broad class of objects in the same way. We base our method on a generic skeleton concept to describe the geometry, completed by specific architectural parameters according to the studied shape.

3.1 Shape parameterization

3.1.1 Geometrical parameterization

We consider the skeleton as a set of curves composed of a generating curve and section curves.

The purpose of the generating curve is to describe the general shape of the object. For airfoil based shapes, the trailing edge is an ideal choice, as is the keeline for a hull.

The sections are similar to the classic architects line plan, describing more precisely the outlines of the object around the generating curve. Each section curve is identified on the generating curve by a local coordinate system, an origin and a rotation, allowing to know its position and orientation. Section curves are computed as the intersection between the studied object and a family of cutting planes. The cutting planes are defined to be normal to the tangent vector of the generating curve at the corresponding point adjusted with the rotation associated to the section.

The generating curve and the section curves are represented as B-Splines curves with a given number of control points [1]. To create those curves we use a fitting process, inspired by [11] using a small number of control points (e.g. \( \leq 10 \)). A good level of approximation of the original model is ensured, the average normalized relative distance between the intersection curves and the B-Spline section curves is kept under \( 10^{-5} \).

Fig. 1 illustrate the skeleton of the AC45 foil. We illustrate also the skeleton obtained for the hull of a sail boat.
3.1.2 Architectural parameterization

We define a set of architectural parameters on the studied object according to the design practice and effects on the object performance. The strategy of our modeller is to control the whole shape through these parameters.

Both the generating curve and the section curves have an independent set of parameters. Parameters of the foil are illustrated in Fig 2.

We introduce an observer function $\phi$ that computes the set of parameters $P$ on a given geometry $G$: $\phi : G \rightarrow P$. For the generating curve the parameters are real and finite values whereas sections describe parameters as a function along the generating curve, thus defining $\phi$ in an infinite dimensional space.

In practice $\phi$ is represented with B-Spline curves $B_\phi$ passing through the section parameter values according to their position on the generating curve.

The control points of $B_\phi$ will be used to control the value of the sections parameters. The number of control points of $B_\phi$ is chosen to be way smaller than the total number of sections of the skeleton. This allows drastically reducing the number of parameters that control the shape of the object and that are used in an optimization loop. In addition, the modification of a B-Spline curve can ensure a smooth distribution of the parameters, preserving the fairness of the object.
3.2 Shape deformation

Our goal is to compute smooth shapes corresponding to a given set of architectural parameters. Therefore, deforming an object corresponds to finding a new geometry $G$ that matches a given set of architectural parameters $P$. Referring to the definition of the observer function $\phi$, we need to compute an inverse problem: $\phi^* : P \rightarrow G$.

The shape $G$ is described with a skeleton made of B-Spline curves. The idea is to compute new values of the coordinates of B-Spline curves control points until the new skeleton parameters reach the target ones. The new coordinates of the B-Spline control points are the solution of a non-linear constrained minimization problem, built with four terms, as described below.

1. $E_{\text{param}}$ measures the distance between the current parameters values and the target ones.
2. $E_{\text{shape}}$ is introduced to ensure consistency control by measuring the distance between the current geometry and the original one.
3. The third term allows taking into account specific constraints $F$ for the studied object, usually position or tangency constraints. These constraints are defined for each section and are not necessarily the same for all sections.
   For example, an airfoil has a smooth connection between the suction and pressure faces thanks to a tangency constraint: the tangent at the leading edge of both sides has to be orthogonal to the chord vector.
4. The last term $H$ controls the overall smoothness of the shape by introducing stiffness between successive control points of the section or generating curves. We add correction terms to control respectively $C^1$ and $C^2$ properties of control points.

The proposed minimization system is:

$$\min_{c_i} E_{\text{param}} + \varepsilon E_{\text{shape}} + \sum_k \lambda_k P_k^2(c_i) + \sum_l \mu_l H_l(c_i)$$  \hspace{1cm} (1)

where $c_i$ represents the control points of the generating curve or a given section of the skeleton. The system (1) is solved for the generating curve and for each section curve independently.

The definition of the problem (1) is well adapted to Sequential Quadratic Programming (SQP). We start with an initial value of $\varepsilon$, usually 1, and the original curve as the starting point of the algorithm, then we decrease $\varepsilon$ at each iteration and start the SQP again with the last computed curve. The algorithm stops when the value of the objective function reaches a fixed threshold. $\lambda_k$ and $\mu_l$ are chosen small, usually $10^{-4}$.

4 AC45 FOIL MULTI-OBJECTIVE SHAPE OPTIMIZATION

In the recent years, new high-speed boats were developed using foils. The purpose of a foil is to lift the hull of the boat above water surface. The hull resistance (friction and wave making
For sailing yachts, the foils are built as an "L" shape with a vertical part countering the sails forces, and a horizontal part supporting the yacht weight. While sailing, the foil allows the yacht to fly as shown in Fig. 3. However, to maintain this flying state, the stability of the foil is a critical aspect for both safety and performance. Designers have to manage numerous parameters in order to produce a foil with a low drag, but high stability.

We consider here the AC45 foil. This type of foil is "one-design" meaning that its shape is the same for all AC45 boats. For this application, we aim to optimize the shape of the AC45 foil in order to decrease its total drag while keeping stability as high as possible. The foil performances are computed with the potential flow solver AVANTI coupled with Xfoil.

The AC45 foil is currently used by the Groupama Team France sailing team for the 35th America’s Cup. In such a context, the performance of the foil is essential. An illustration of the sail boat flying thanks to the foil is shown in Fig. 3, one foil in the water (right) and the other one visible in the retracted position (left).

4.1 Numerical methods

4.1.1 AVANTI

AVANTI, the flow code used in the present study, is developed and commercialized by the company K-Epsilon. AVANTI features multiple different methods for flow solving (e.g. vortex line method, particle method, panel method, etc.) [12, 13].

The method used here is a vortex line method with solved wake. AVANTI is coupled to XFOil in order to incorporate the flow behaviour such as laminar transition, and stall.

The foil is represented with a finite number of elements, i.e. airfoil sections given by the skeleton. For each element a local velocity, a local Reynolds number and a local angle of attack is computed. Each element has an associated XFOil database containing the lift and drag of the section for a given range of angles of attack.

AVANTI uses this database to find the lift of each element of the foil according to its current
local angle of attack. Then the lift is converted to a local vorticity. The wake is imposed with the computed gradient of vorticity then solved. These steps are repeated until convergence thanks to a direct iterative method, which is able to find a stationary solution.

As inputs, AVANTI requires a 2D point cloud description of the sections with its the 3D position given with points and quaternions. These files are generated automatically by the parametric modeller.

In our specific case for AC45 foil study, only the underwater part of the foil is simulated. The influence of the free surface is taken into account with an anti-symmetry plane model. This model is a satisfying approximation for high speed. As [14] suggests, with a Froude number greater than 1, an infinite Froude number free-surface condition can be used. In our case, the Froude number is around 5.45.

We illustrate in Fig. 4 the wake computed with AVANTI and the vortex lines. The vortex line is located at 25% of the aft of the leading edge along the foil. From the vorticity repartition colormap, we see that the parts of the AC45 which generate most of the force allowing to lift up the boat are the knee and the tip.

The reference frame is defined as follows: $X$ is in the opposite direction of the flow, $Z$ is in the vertical direction (oriented upwards) and $Y$ is horizontal, perpendicular to $X$.

4.1.2 FAMOSA

The robustness of the optimization algorithm is critical to solve realistic problems. Therefore, derivative-free methods have been preferred to more efficient but fragile gradient-based approaches.

Evolution strategies mimic the natural evolution laws to simulate a population of individuals that progressively converges to the optimum. In this paradigm, each individual is characterized by a set of parameters and its ability to survive is proportional to its performance. These methods, although expensive, are noticeable since they are able to avoid local minima thanks to random operators.
FAMOSA toolbox features the PAES algorithm [15], which is a particular evolution strategy, which has been adapted to the context of multi-criterion optimization. In that case, the ability of an individual to survive is not related directly to the criteria values, but to the concept of dominance, introduced by the economist Pareto. In short, an individual is dominating another one if it has a better performance according to all criteria. This algorithm generates an archive of non-dominated individuals, which is used to determine the ability of a new individual to survive and have offsprings.

4.2 Performance criteria

We choose to define the foil performances with two criteria computed with ARAVANTI.

1. The total drag $F_x$ of the foil in the reference frame. A low drag increases the total performance and speed of the boat.

2. A stability criterion, represented by $\frac{\partial F_z}{\partial z}$, where $F_z$ is the total force in the $z$ direction of the foil. The aim of this criterion is to ensure that the boat will stay at a fixed $z$ height thanks to a self adjusting $F_z$ balancing the vertical movements of the foil.

Computations are performed with a fixed $F_y$ given as the opposite force to balance the force applied by the sails on the hull. $F_z$ is also fixed to counter the weight of the hull and be able to lift it up. The speed of the yacht is set to 22 knots. ARAVANTI solves for the leeway and rake angles of the foil, until computed forces converge to the imposed forces.

$F_x$ is computed during the simulation, and we aim to decrease it as much as possible. In the reference frame we used, $F_x$ is oriented along the negative $x$ direction. Thus, the sign of $F_x$ will be negative, but we can consider the absolute value to compare the foil performance.

To compute the second criterion, we estimate $\frac{\partial F_z}{\partial z}$ with finite differences. We vary the foil displacement by a small $\Delta z$ and compare it to the computed $F_z$. To be stable, the foil has to generate a $F_z$ opposed to the direction of the displacement. Thus the ratio $\frac{\partial F_z}{\partial z}$ has to be negative and as large as possible. For example, if the boat is riding too high above the water surface, the foil force $F_z$ has to decrease in order to make the whole system lower.

The aim of our study is to reduce the total drag of the AC45 as much as possible while keeping stability criterion as large as possible.

4.3 Shape parameters

Deformation of the foil shape is decomposed into general form parameters (generating curve parameters) and local form parameters (section curve parameters), as illustrated in Fig. 2. We have identified the most relevant parameters that influence a foil performances as: the tip length, the angle between the shaft and the tip, the cant angle, the local chord and twist of sections. Five parameters are used to control the chord along the foil, and five others to control the twist.
These parameters are obtained with the observer function representation $B_\phi$ described in the section 3.1.2.

To generate a new CAD from the original CAD model, our tool takes on average 12 seconds to build the skeleton, 5.1 second for the generating curve deformation and 5 seconds for the section curves deformation. There is no need to build a new surface around the skeleton, as AVANTI does not require a continuous surface as an input. On average, AVANTI takes 20 seconds to perform the computation of the two criteria and post-process the result.

The PAES algorithm does not require limits to parameters variations. A starting point and an initial step length is defined to explore the domain. Then, the search direction is oriented towards the best results found, until the algorithm converges to the Pareto front.

4.4 Results

We illustrate in Fig. 5 the Pareto front obtained. The performances of the AC45 are identified with the orange triangle. The blue points represent an initial distribution of shape parameters along a domain around the initial shape of the AC45. The distribution follows a Latin Hyper cube model. The green points represent the path of the PAES algorithm to converge towards the Pareto front. Finally, the red points are located on the Pareto front.

On the Pareto front we identified 4 foils labelled F1, F2, F3, F4 illustrated in the Fig. 6 to 9. The resulting shapes demonstrate the capability of the modeller tool to generate geometrically valid forms. The shapes of the foils on the Pareto front tends towards much thinner forms than the original AC45 foil to limit drag. The tip length is longer, in order to counterbalance the loss of lift induced by smaller airfoils. Larger cant angles observed in all four foils improves the stability criterion without impacting the drag. We can notice that the upper part of the foil, above the water surface, is not modified by the shape optimization. Indeed, this part of the foil does not impact significantly the performance criteria.

The structural properties of the four foils can be discussed: such thin shapes can lead to structural deficiencies. Moreover, we did not introduce a criterion to avoid cavitation phenomenon. This study highlight the importance of multi-physics optimization. Taking into account structural properties of the shapes will lead to more feasible shapes. The current results are relevant according to the geometric and hydrodynamic criteria chosen.

5 CONCLUSION

This paper presents a method for smooth shape deformation with a generic skeleton-based approach. The twofold parametrization, geometrical and architectural, demonstrates its capability to generate simulation-suited models. Our parametric modeller allows to explore the domain of possible shapes in an efficient way and to determine improvements of the design that are architecturally relevant.

As shown by the experiments, we are able to improve the hydrodynamic performances of a AC45 foil in an efficient and automatic way.
We also implemented a technique to reconstruct with accuracy a 3D surface around the deformed skeleton that we did not describe here. With this feature, the parametric modeller can also be linked with different types of flow or structural solvers.

Further work will focus on including multi-physics criteria in the optimization loop. We will also focus on handling more complex geometries with the skeleton representation. Section curves with multiple components and branching curves will be possible.

Acknowledgements
The project was achieved with the financial support of ANRT (Association Nationale de la Recherche et de la Technologie).

REFERENCES
metric hull form definition on hydrodynamic performance optimization. 6th International Conference on Computation Methods in Marine Engineering (2015)


Figure 6: Views of the foil labelled $F1$

Figure 7: Views of the foil labelled $F2$

Figure 8: Views of the foil labelled $F3$

Figure 9: Views of the foil labelled $F4$
LONG-TERM PROBABILITY DISTRIBUTION OF FIXED OFFSHORE STRUCTURAL RESPONSE USING AN IMPROVED VERSION OFFINITE MEMORY NONLINEAR SYSTEM PROCEDURE

N.I. MOHD ZAKI*, M.K. ABU HUSAINa AND G. NAJAFIANb

*UniversitiTeknologi Malaysia, Jalan Sultan Yahya Petra, 54100 Kuala Lumpur, Malaysia
e-mail: noorirza.kl@utm.my

aUniversitiTeknologi Malaysia, Jalan Sultan Yahya Petra, 54100 Kuala Lumpur, Malaysia
e-mail: mohdkhairi.kl@utm.my

bSchool of Engineering, University of Liverpool, L69 3GQ, United Kingdom
e-mail: najafian@liv.ac.uk

Abstract. Offshore structures are exposed to random wave loading in the ocean environment and hence the probability distribution of the extreme values of their response to wave loading is required for their safe and economical design. Due to nonlinearity of the drag component of Morison’s wave loading and also due to intermittency of wave loading on members in the splash zone, the response is often non-Gaussian [1-2]; therefore, simple techniques for derivation of the probability distribution of extreme responses are not available. However, it has recently been shown that the short-term response of an offshore structure exposed to Morison wave loading can be approximated by the response of an equivalent finite-memory nonlinear system (FMNS) [3]. Previous investigation shows that the developed FMNS models reduce the computational effort but the predictions are not very good for low intensity sea states. Therefore, to overcome this deficiency, a modified version of FMNS models is referred to as MFMNS models is used to determine the extreme response values which improves the accuracy but is computationally less efficient than FMNS models. In this paper, the 100-year responses derived from the long-term probability distribution of the extreme responses from MFMNS and FMNS models are compared with corresponding distributions from the CTS method is investigated with the effect of current to establish their level of accuracy. The methodology for derivation of the long-term distribution of extreme responses (and the evaluation of 100-year responses) is discussed. The accuracy of the predictions of the 100-year responses from MFMNS and FMNS models will then be investigated.

1 INTRODUCTION

For offshore structural design, the load due to wind-generated random waves is usually the most important source of loading. Whilst these structures can be designed by exposing them to extreme regular waves (100-year design wave), it is much more satisfactory to use a probabilistic approach to account for the inherent randomness of the wave loading [1]. This approach allows the statistical properties of the loads and structural responses to be determined, which is essential for risk-based assessment of these structures. The major
obstacle in achieving this objective is the nonlinearity of the wave load mechanism resulting in non-Gaussian response distributions [2], due to the drag component of Morison wave loading. Furthermore, dynamic effects, the presence of current and load intermittency in the splash zone, all have an effect on the statistical properties of structural response [3, 4], increasing the complexity of the problem.

Many different techniques [5] have been introduced for evaluation of statistical properties of offshore structural response. Examples of time domain techniques include (standard or conventional) Monte Carlo time simulation, Finite-memory nonlinear system and NewWave theory. Also, in the frequency domain, the Volterra series has been used to calculate the skewness and kurtosis of structural response from its higher order spectra (bi-spectra and tri-spectra). However, in the most part, these methods have major shortcomings such as their inability to account for current or load intermittency, and/or are limited to quasi-static responses or to very simple structures with a few nodal loads.

In reality, Monte Carlo time simulation is the most reliable technique in that it could readily account for all forms of nonlinearities [3]. In this technique, a random wave record is first simulated from a given frequency spectrum (representing a particular sea state), and a corresponding response record such as a base shear record is then simulated. The extreme value of this record will be a sample extreme response. The process is then repeated many times to have a large sample of extreme responses, which will then be fitted to a suitable extreme value probability distribution model. However, very large samples (tens of thousands) are required to prevent excessive sampling variability [6]. Naturally, this results in an unacceptably high computational cost, especially as structures become more complex and increasingly economical designs are required.

A new technique has recently been introduced [1] by using the mean upcrossing rate function in predicting extreme response values. The results of the simulation were optimized with linear extrapolation for more accurate prediction. Najafian [3] applied the same technique but proposed a finite-memory nonlinear system (FMNS) model to increase efficiency. Based on the validation test, the model was able to improve efficiency, but it was less accurate as compared to Monte Carlo time simulation. Hence, Najafian and Mohd Zaki [7] improvised the model that is able to improve both efficiency and accuracy. However, the model is only relevant for structures under high significant wave height ($H_s$) values.

With further investigation, Mohd Zaki et al. [8] introduced a method to increase the accuracy on low $H_s$ values by dividing the structure into two parts. Each leg of structure is modelled separately using the FMNS model and the total extreme response will be the summation of the extreme responses in each part. This is known as a modified finite-memory nonlinear system (MFMNS). However, the method is still limited only to certain sea state conditions, such as the presence of currents along wave propagations.
2 CONVENTIONAL TIME SIMULATION (CTS) PROCEDURE FOR SIMULATION OF RESPONSES

Conventional time simulation (CTS) technique has been used in this study for the following two reasons: a) to identify system B of the FMNS model (refer to Figure 1), and b) to establish the accuracy of the FMNS models by comparing the probabilistic properties of the response from FMNS models with those from the CTS method. To identify system B, a long record (about 4.5 hours in this study) of both nonlinear and its associated linear quasi-static responses were simulated; hence in this section the simulation of both nonlinear and linear quasi-static responses are discussed.

The procedure of simulating a sample of response record is summarized as follows:

1. Identify the appropriate frequency wave spectrum (i.e. Pierson-Moskowitz spectrum) based on the location of the offshore structure, provided the significant wave height, \( H_s \) value and its corresponding \( T_z \) value.

2. Generate surface elevation based on the appropriate frequency wave spectrum at an arbitrary reference point for a given period of time. The surface elevation at point, \( x \) and time, \( t \) can then be expressed as:

\[
\eta(x, t) = \sum_{i=1}^{NW} A_i \cos(2\pi f_i t - k_i x - \varphi_i)
\]

where \( NW \) is the total number of wavelets used in the simulation, \( f_i \) are a set of equally-spaced discrete wave frequencies, \( k_i \) are their associated wave numbers, \( \varphi_i \) are random phase angles distributed uniformly in the range \( 0 < \varphi < 2\pi \), and finally \( A_i \) are wave amplitudes.

3. Compute the component of water particle kinematics (velocities and accelerations) according to linear random wave theory at each node elevation using the appropriate transfer function and account for the intermittency load at the member of the splash zone as defined below.

\[
\Gamma_u(f_i) = \omega_i \frac{\cosh(k_i z)}{\sinh(k_i d)} e^{-j k_i (x-x_0)}
\]

\[
\Gamma_a(f_i) = j \omega_i \Gamma_u(f_i), \quad j = \sqrt{-1}
\]

where \( \Gamma_u \) and \( \Gamma_a \) are the transfer functions for water particles kinematics; velocity and acceleration, respectively at a particular nodal point, in which \( x \) is the horizontal coordinate of the node and \( x_0 \) is the coordinate of arbitrary reference point where the surface elevation has been simulated. \( \omega \) stands for angular frequency where \( \omega_i = 2\pi f_i \), \( k \) is their corresponding wave numbers, \( d \) is the water depth that indicates vertical distance.
between mean water level (MWL) and the seabed and \( z \) refer to elevation above the seabed. The vertical stretching approach will be applied for the condition above the mean water level.

4. Compute the Morison load at different nodes corresponding to its water particle kinematics. That is,

\[
F = F_d + F_i = k_d u|u| + k_i \ddot{u}
\]

where \( F_d \) and \( F_i \) are the drag and inertial components of fluid loading; \( C_d \) and \( C_m \) are empirical drag and inertia coefficients; \( D \) is the cylinder diameter; \( \rho \) is the water density; \( u \) and \( \ddot{u} \) are the horizontal component of water particle velocity and acceleration at the centre of the cylinder. For a more complete review of Morison’s equation refer to Sarpkaya and Isaacson [9].

5. Calculate the drag-induced, \( \vec{R}_d \) and inertia-induced, \( \vec{R}_i \) components of the response (quasi-static base shear and overturning moment in this study). As the structural system is assumed to be linear, the total response would then be equal to the sum of the foregoing two components.

3 FINITE-MEMORY NONLINEAR SYSTEMS

Finite-memory nonlinear systems, as depicted in Figure 1, are extensively used in system identification techniques to establish a relationship between the output and input of some nonlinear systems [3]. In the case of a jacket structure exposed to random waves, the input is the surface elevation at a reference point and the output is the dynamic response. The following is meant to briefly explain how a finite-memory nonlinear system can be used to establish a simple relationship between the output and input of a jacket structure. For a more complete description refer to [3, 7].

Figure 1: The structure of Finite-memory nonlinear system.

1. The water particle kinematics at different nodes of the structure are calculated based on linear random wave theory by applying appropriate transfer functions to the surface elevation at a given point, \( \eta(t) \). On the other hand, the linear quasi-static response (denoted by \( \vec{R}_i(t) \), where \( R \) stands for response; the symbol \( \sim \) on top of \( R \) indicates quasi-static; and finally, the subscript \( L \) stands for linear) is equal to a linear combination of linearised Morison loads acting at different nodes. Since the linearised Morison loads are
themselves linear combinations of their corresponding water particle kinematics, it can be concluded that the linear quasi-static response is linearly related to the surface elevation, via a transfer function. This linear process is represented by the first linear system (system A) in Figure 1. In other words, system A converts the surface elevation at a reference point to the linear quasi-static response.

2. The next stage is the calculation of approximate values of the (nonlinear) quasi-static response (i.e. \( \hat{R} \) where the symbol \( \sim \) on top of \( R \) stands for approximate value) from their corresponding linear response values through a memoryless nonlinear transformation. That is, instantaneous values of quasi-static response are assumed to depend on the values of the linear quasi-static response at the same instant. This process is represented by the nonlinear system B in Figure 1. System B accounts for nonlinearities due to both drag component of Morison loading and also nonlinearities due to load intermittency on members in the splash zone.

3. The final stage is the calculation of approximate (dynamic) response, from its corresponding (approximate) quasi-static response through a suitable transfer function. This process is represented by the linear system C in Figure 1. This stage is quite straightforward as the appropriate transfer functions (frequency response functions) are determined from established procedures in modal analysis of linear structures.

3.1 Modified Finite-Memory Nonlinear System (MFMNS)

Current finding shows that the modified finite-memory nonlinear system (MFMNS) technique able to determine the short-term probability distribution of response extreme values efficiently [8]. This modified version of FMNS performs better in terms of accuracy. The poorer performance of standard FMNS, particularly for low \( H_s \) values, can be improved by dividing the structure into two parts (parts 1 and 2) in order to minimize the horizontal distance between nodes in each part compared to the wavelengths.

By applying systems A and B (refer to Figure 1), the quasi-static response is modelled separately according to the part. Then, the quasi-static responses at each part need to be combined to obtain the total quasi-static response. In this paper, the waves are assumed to propagate in the global Y direction. Therefore, the first two columns in a plane perpendicular to the wave direction (XZ plane) constitute the first part, with the remaining two columns belong to the second part (refer to Figure 1). Note that the standard FMNS was developed analogous to a single column. In effect, long waves perceive the structure as a single column.

4 LONG-TERM PROBABILITY DISTRIBUTION OF EXTREME RESPONSES

In this section, the long-term probability distribution of the extreme responses from MFMNS models are compared with corresponding distributions from CTS method to investigate the level of accuracy of the MFMNS models in predicting the probability distribution of the extreme responses. The long-term distribution of extreme responses can be derived by convoluting the short-term distribution of extreme values by the long-term distribution of sea states. That is,
where \( p(H_s, T_z) \) is the joint probability density function of significant waveheight and mean-zero upcrossing period. Using the extended scatter diagram as an approximation for the joint probability density function, the above equation can be written as [10],

\[
P_{lt, r_{\text{max}}} (q) = \sum_i \sum_j P_{r_{\text{max}}} (q | H_{si}, T_{zi}) * \frac{W_{ij}}{W}
\]

where \( W_{ij} \) is the number of occurrences of the sea states represented by \( H_{si} \) and \( T_{zi} \) in the scatter diagram and \( W = \sum_i \sum_j W_{ij} \) is the total number of sea states. In other words, \( W_{ij}/W \) is the probability of occurrence of the sea state characterised by \( H_{si} \) and \( T_{zi} \). Meanwhile, \( P_{r_{\text{max}}} (q | H_{si}, T_{zi}) \) is the short-term probability distribution of response extreme values for the sea state characterised by \( H_{si} \) and \( T_{zi} \).

In this study, for each sea state, the distribution of extreme values is calculated based on 1000 simulated records, each of 128sec duration. In order to calculate \( P_{lt, r_{\text{max}}} (q) \) from Eq. (6), the values of \( P_{r_{\text{max}}} (q | H_{si}, T_{zi}) \) must be known for all simulated \( q \) values belonging to all sea states. \( P_{r_{\text{max}}} (q_n | H_{si}, T_{zi}) \) is calculated in the following way. Rank all the simulated extreme values for a given sea state from smallest to largest. Then assuming that the number of simulated extreme values for each sea state is \( N \) (\( N = 1000 \) in this study) and assuming that \( q_n \) is the \( n \)th smallest simulated extreme value, use the following equations to calculate the short-term distributions

\[
P_{r_{\text{max}}} (q_n | H_{si}, T_{zi}) = \frac{n - 0.44}{N + 0.12}
\]

and

\[
P_{r_{\text{max}}} (q < q_1 | H_{si}, T_{zi}) = 0
\]
\[
P_{r_{\text{max}}} (q > q_N | H_{si}, T_{zi}) = 1
\]
\[
P_{r_{\text{max}}} (q_1 < q < q_N | H_{si}, T_{zi}) = \text{determine by the interpolation from Eq. (7)}
\]
Furthermore, the method of moments was used to fit a Gumbel distribution to the simulated extreme values for each sea state. The long-term distributions of extreme values are then calculated by convoluting the short-term Gumbel distributions of extreme values with the probability of occurrence of each sea states (Eq. (7)). The foregoing long-term distributions are then used to calculate the 100-year responses for CTS, FMNS and MFMNS methods.

4.1 Derivation of the 100-Year Extreme Response from the Scatter Diagram

The 100-year extreme response is defined as the extreme response which, on average, is equalled or exceeded once every 100 years. Now, the number of 128-sec intervals in 100 years is \( N = \frac{100 \times 365 \times 24 \times 3600}{128} \). Therefore, on average, only in one 128-sec interval out of \( N \) intervals, the extreme response would be higher than \( R_{100} \). (In other words, the extreme response in \( (N-1) \) intervals must be less than \( R_{100} \)). Then, the probability of the extreme response in a 128-sec interval being larger than \( R_{100} \) would be \( \frac{1}{N} \). That is,

\[
\text{Prob}\{\text{extreme response} > R_{100}\} = \frac{1}{N} \quad \text{or} \quad \frac{1}{100 \times 365 \times 24 \times 3600/128} = 4.0589 \times 10^{-8}
\]

Knowing the probability of exceedence, its corresponding 100-year response can be calculated from the long-term distribution of extreme responses (based on Gumbel distributions fitted to simulated extreme responses for each sea state).

5 TEST STRUCTURE AND RESPONSES

The preceding sections demonstrated the procedure of evaluating offshore structural responses based on the appropriate wave theory. The evaluation was applied to the test structure that is similar to a fixed jack-up platform by applying the drag and inertia coefficient according to API RP2A-WSD standard code (2007) of \( C_D = 1.05 \) and \( C_m = 1.20 \). As shown in Figure 2, the test structure is a quasi-static fixed platform consisting of four vertical legs, each with a 1.5m diameter and wall thickness of 40mm, located in a water depth of 110m. The legs support a 35m x 38m platform deck and the hydrodynamic load is distributed along 30 points on each leg. The foregoing test structures were subjected to various uni-directional sea-states simulated from Pierson–Moskowitz (P–M) frequency spectrum. The responses chosen for investigation were the base shear and overturning moment of the test structure.

Adjustment of the Young’s Modulus of the bracing elements of the structure (not shown in the figure) allows control of the overall stiffness and hence the natural frequency of the structure (the internal bracings were assumed to receive no fluid loading to reduce the computational effort). JCP2, JCP5 and JCP8 are used to refer to three FE models with first mode natural periods of 2.53, 5.21 and 8.12 seconds, respectively. For a more complete description of the test structure refer to [5].
6 COMPARISON OF 100-YEAR RESPONSES FROM MFMNS AND CTS METHODS

The long-term probability distributions of the maxima for the case of (total) quasi-static base shear with zero current are shown in Figure 3. As observed, there is good agreement between the distributions from CTS and MFMNS methods (both simulated and Gumbel-based distributions). The long-term probability distributions of the maxima for JCP5 base shear with zero current, and for JCP2 base shear with negative current, are shown in Figures 4 and 5, respectively, and similar conclusions can be made. It is observed that the 100-year extreme response for all cases are predicted with very good accuracy with maximum inaccuracies of 6% for negative current case.

It is also observed that the 100-year maxima are very significantly underestimated by linear modelling of the response. The worst case is that of the JCP5 overturning moment with zero current with a ratio of 0.40 between linear and nonlinear 100-year maxima. The 100-year minima are also significantly underestimated by the linear response model in most cases. The
only exception is that of the positive current, where due to extreme asymmetry, the linear 100-year minima are sometimes higher than their corresponding nonlinear values.

Figure 3: Comparison of 100-year (maxima) from CTS and MFMNS methods. 1000 sample records for each sea state, $T = 128$ sec. Total base shear, quasi-static, current = 0.00 m/sec.

Figure 4: Comparison of 100-year (maxima) from CTS and MFMNS methods. 1000 sample records for each sea state, $T = 128$ sec. Total base shear, JCP5, current = 0.00 m/sec.
7 CONCLUSIONS

- Two different versions of finite-memory nonlinear systems for modelling offshore structural response due to Morison's wave loading have been reviewed: FMNS and MFMNS. It was concluded that the accuracy of FMNS, is poor for low $H_s$ values ($H_s = 5m$), and that MFMNS is more appropriate.

- The level of accuracy of MFMNS/FMNS models is established by calculating the ratios between the 100-year responses from the MFMNS/FMNS methods and corresponding values from the conventional time simulation method (CTS). The maximum inaccuracies in predicting the 100-year responses from MFMNS and FMNS methods are 3% and 7%, respectively. So while MFMNS is more accurate, the accuracy of the FMNS is also good.

- Finally, it has also been confirmed that the 100-year maxima are very significantly underestimated by linear modelling of the response with ratios as low as 0.40 between linear and nonlinear 100-year maxima. Therefore, linearization can lead to significant under design of the structure with the very high risk of structural failure when the structure is exposed to a severe sea state.

REFERENCES


MULTICOMPONENT DESIGN OF ROTOR-STATOR-NOZZLE (RSN) PROPULSOR ON AZIPODS

MARINE 2017

NIKOLAY V. MARINICH*, ALEKSEY YU. YAKOVLEV*, NIKOLAY A. OVCHINNIKOV* AND TOMI VEIKONHEIMO†

* Krylov State Research Centre
196158, St. Petersburg, Russia, 44, Moskovskoye shosse
e-mail: 10_otd@ksrc.ru, www.krylov-center.ru
† ABB Marine and Ports
P.O.Box 185, Merenkulkjankatu 1,FI-00981 Helsinki, Finland
e-mail: Tomi.Veikonheimo@fi.abb.com, www.abb.com/marine

Key words: Nozzled rotor, Post-swirl stator, Computational Methods, Design, Optimization, CFD calculation, Experiment

Abstract. The paper offers design computation method of RSN propulsor fitted with post-swirl stator on AZIPODs. The method implemented as problem solution of non-linear optimization of objective functional with restrictions over the field of infinitely dimensional values (functions being sought for). Numerical solution of this problem is reduced to the finite dimensional problem. The shape of internal and external nozzle surface, distribution of pitch, camber, width and thickness of rotor blades and post-swirl stator geometry at radius are unknown functions of RSN propulsor design. Computational accuracy was validated through CFD-aided calculation of designed propulsors. Comparative analysis with model test results was implemented as well. The AZIPOD XL propulsor concept has been developed by the present method.

1 INTRODUCTION

Heavily loaded propellers are fitted with nozzles since the first half of the XXth century. In the extreme case - bollard pull, mainly typical for tugs, nozzle allows doubling the propulsion thrust. Thrust increase at relatively low speed is also important for other kinds of ships, e.g. fishing, research, and support vessels. Tough positioning requirements for special-purpose vessels called into existence nozzled pods. Along with these propulsors, there also exist nozzled propellers for fast ships, see [1]. In a number of cases, nozzled propulsors have stator blades installed upstream or downstream of the propeller, see [1], [2]. Traditionally, nozzled propeller design is based on the diagrams of test series, see [3]. Later on, various methods of design calculation were used for propeller design, see [4], [5], [6], the nozzle shape normally remaining the same. These typical nozzle shapes are well known: D19a or more advanced ones, as per [7] and Soviet-developed OST nozzles [3].
Although there did appear a number of investigations on nozzle shape selection, e.g. [8], [9]. To some extent, integrated design of nozzled rotors was discussed in [10], [11], [12].

The approach to propulsor design must be holistic, so the potential of conventional design calculation methods cannot be used to the full. In this case, the methods based on direct numerical solution to the optimization problem are a universal tool. For nozzled rotors, this approach is used in [13].

2 PROBLEM STATEMENT

In view of the above, the purpose of this paper is to develop a design method for ducted rotor with post-swirl stator (RSN) (Fig. 1) as a totality of interwoven elements, that would yield better target parameters (efficiency, thrust) than those of conventional propulsor designs. This design method is based on direct optimization of propulsor components, and along with it, it should be fast and handy. The latter requirement determined the choice of BEM-methods with simplified consideration of viscous effects at the direct design stage and application of RANS-codes in further verification calculation.

[Figure 1: RSN propulsor (View of designed unite).]

This design methods is based on the studies performed by Krylov State Research Centre, including vortex theory-based methods for verification of nozzled rotor calculations and Boundary Element Methods [14], design calculation method for nozzled propeller based on direct optimization, see [15] and [16], analysis of nozzle shape effect upon propulsor performance, see [9], [17] nozzle design method considering viscous effects, see [18]. All these studies made it possible to start the development of an integrated design method for nozzled rotors.

Nozzled rotors have a wide variety of applications, so it has to be determined what elements a nozzled propulsor will consist of and which of them require optimization. This paper discusses RSN propulsors with a single propeller, ring-shaped nozzle, axially symmetric hub and the stator blades downstream the propeller. The propulsor are mounted on the poded unite. All the components mentioned above are optimizable, however the pod shape was fixed.
3 METHOD OF NOZZLED PROPULSOR DESIGN

3.1 Propulsor design algorithm

The propulsor will be designed to the assumed power of the power plant. Once power, RPM and diameter are specified, it becomes possible to determine propeller torque coefficient $K_0^*$ that will also be regarded as pre-defined. The mode of propulsor operation will depend on the propeller thrust and the force on post-swirl stator, so it may vary during design process.

The propulsor consists of several elements with completely different operation principles, so the design process is iterative, all the elements being designed one by one at each iteration stage. The fist calculation step is to design the propeller to the specified torque coefficient, the nozzle geometry being regarded as fixed. To optimize the propeller, it is necessary to specify inner advance ratio $J_S$ calculated for the flow rate $Q$ at the nozzle in the section of the propeller disk.

$$J_S = \frac{Q}{\pi n D^3}$$ (1)

Here, $n$ – rotation rate of rotor, $D$ – rotor diameter.

To find this value, successive iterations are required because in the optimization to the specified torque coefficient, thrust coefficient $K_T$ that determines flow rate in the nozzle, only becomes known after the verification calculation is done. For each new propeller geometry, the stator is designed accordingly, and the optimal shape of nozzle is determined.

As nozzle optimization proceeds, the shape of the nozzle changes significantly, which has a strong effect upon the inner advance ratio. The nozzle also makes axial velocity distribution along the propeller disk radius non-uniform, which means the change in the operational conditions of the propeller, so one more iteration cycle is required. After several iterations, the process of propulsor optimization converges to the final solution.

3.2 Unified approach to optimization of specific propulsor components

As it was shown above, the algorithm mainly concerns optimization of specific propulsor elements. This optimization is based on a unified approach. Mathematically, determination of optimal shape for a body can be represented, in the most general form, as finding the minimum of a certain functional $I$, so as to satisfy a number of restrictions $G_i$, i.e.

$$\{I(\varphi_j(x),...)\to\min$$

$$G_i(\varphi_j(x),...)\leq0, i=1,n$$

(2)

Here, $\varphi_j$ are target functions, and their number can be arbitrary.

Expressions for $I$ and $G_i$ may vary significantly, depending on the given task. At the same time, functions $\varphi_j$ involved in Expression (2) describe the shape of the body in the flow, so they must meet a number of requirements, including the requirements of continuity and piece-wise differentiability. On the other hand, $I$ and $G_i$ are usually found numerically, by means of boundary integral equation methods (BEM), and in future they could be obtained by means of RANS-methods, too. Accordingly, $I$ and $G_i$ can be regarded as smooth, but not as differentiable.
Problem (2) is reduced to mathematical programming problem [19]. The problem of mathematical programming is formulated in the space of N-dimensional vectors, where N can be high but not infinite. Accordingly, the task is to discretize Problem (2), i.e. to pass from the real functions forming infinitely-dimensional space to the finite number of target parameters. This can be done by representing target function \( \varphi_j \) as a linear combination of basis functions \( f_k \).

\[
\varphi_j(x) = \sum_{k=1}^{N} A_{kj} \cdot f_k(x)
\]  

Selection of basis functions depends on the specific task. From the narration above, it is clear that they must be continuous and have at least the first-order derivative continuous. The system of basis functions is generally selected so that they were orthogonal.

With expression (3) inserted into expression (2), optimization problem can be formulated as:

\[
\begin{align*}
\hat{I}(\hat{x}) & \rightarrow \min \\
\hat{G}_i(\hat{x}) & \leq 0, i = 1, n
\end{align*}
\]  

Here, \( I \) represents the function of n-dimensional vector \( A \) generated by target coefficients \( A_{kj} \) in Expression (3), for all functions \( \varphi_j \).

\[
\hat{A} \in S \subset R^N, N = \sum_j N_j
\]  

Restrictions of problem \( \hat{G}_i \) are also functions of N-dimensional vector \( A \).

From the calculation viewpoint, Problem (4) is a non-linear problem of mathematical programming with restrictions, see [20]. In this case, the problem is solved by means of Powell method which is a variant of conjugate directions method. Its advantages are that it does not require calculation of derivatives and always converges to the local optimum, see [20].

Today, computers became significantly faster thanks to wide use of parallel calculations. But Powell method, as it is described in [20], is strictly sequential, so it had to be modified accordingly, see [19]. To see what efficiency parallel calculations could show in this case, test calculations were performed with various number of basis functions, and their results confirmed the efficiency of the algorithm based on parallel calculations, see [19].

### 3.3 Rotor optimization

Geometry of propeller blades is determined by several parameters being the functions of radius \( r \) and distance measured along cylindrical cross-section chord \( \xi \), see [3]. Blade width is selected so as to ensure necessary cavitation margin. The thickness is calculated as per the rules of the Classification Society selected. Distribution of blade pitch \( P \) and camber \( f \) over radius \( r \) are considered unknown, as well as blade area ratio \( A_e/A_o \) of the rotor and maximum thickness \( e(r) \) of its blades.

Objective functional in rotor optimization may be represented as:
\[ I = 1 - \eta \left( P(r), f(r), \frac{A_r}{A_o}, \epsilon(r) \right) + k_1 \cdot \sqrt{\sigma(J^*)} + k_2 \cdot \sqrt{\sigma(J^* + \Delta J_1)} + k_3 \cdot \sqrt{\sigma(J^* - \Delta J_2)} \]  

where \( \eta \) is rotor efficiency, \( \sigma \) – cavitation number, \( J^* \) - advance ratio (i.e. ratio between ship speed \( V \) and the nozzle of rotation rate \( n \) and rotor diameter \( D, J = \frac{V}{nD} \) ) in design conditions, \( \Delta J_1, \Delta J_2 \) – desirable offset of cavitation bucket arms from the design advance, \( k_1, k_2, k_3 \) – coefficients ensuring the balance and simultaneously tending to increase efficiency and reduce the risk of cavitation.

Cavitation number \( \sigma \) determines the pressure of cavitation inception on the blade. This number depends not only on the advance ratio but also on the rest of the target values:

\[ \sigma = \sigma(J(r), P(r), f, \frac{A_r}{A_o}, E_{max}) \]  

Selection of \( k_i \) coefficients depends on the requirements the blade system has to meet.

The main restriction \( G_i \) in this problem is that design torque coefficient \( K_Q^* \) be equal to the specified one, \( K_Q \).

\[ G_i = K_Q \left( P(r), f(r), \frac{A_r}{A_o}, E_{max}, J^* \right) - K_Q^* = 0 \]  

Blade area ratio of the rotor is introduced in a standard manner \([21]\) as a ratio between the area of the blades and the area of their circumscribed disk. In this case, the relationship between blade width and blade area ratio is linear. The requirement that this relationship remain linear is the second boundary condition:

\[ G_2 = \frac{A_r}{A_o} - \frac{Z}{\pi R^2} \int_0^R C(r) d r = 0 \]  

where \( Z \) is blade number, \( R \) – rotor radius, \( r_h \) – hub radius.

To obtain an unambiguous solution, blade width distribution along radius \( C(r) \) during optimization keeps its initial shape that must be defined in advance.

The change in the blade area ratio affect not only hydrodynamics of the rotor but its strength, too. To maintain the strength, an additional restriction was introduced, requiring to conserve the section modulus of the blade proportionate to the product of blade width and its square thickness, i.e.

\[ G_3 = C(r) \cdot \epsilon(r) = \Lambda_{avl}(r) \]  

where \( \Lambda_{avl} \) is selected so as to meet strength requirements of the Classification Society. Once this parameter is specified, it becomes possible to design a propulsor to the requirements of various Classification Societies for various classes of ships, depending on specific conditions:

During optimization, rotor parameters have to be recalculated many times, and this is performed as per the method described in \([15]\) and being a variant of vortex method. This method yielded fairly good predictions of rotor parameters.
3.3 Stator optimization

Stator blade geometry is determined by the same parameters as the one of the rotor. Since stator not only improves thrust but also serves as a joint between the nozzle and the pod, width and thickness of stator blades must have sufficient strength margin. Unknown parameters are distributions of blade pitch $H$ and blade curvature $f$ along radius $r$.

Stator is the most efficient only when it fully utilizes the energy of the turbulent wake behind the rotor. Accordingly, the objective functional in rotor optimization must, apart from improving the parameters mentioned above, mitigate the energy of wake vorticity, i.e.

$$\int [V_\theta (r)]^2 \, dr \to 0$$  \hspace{1cm} (11)

where $V_\theta$ is tangential speed behind the stator.

In calculation of both stator and rotor parameters, the same method is applied, see [15], i.e. a variant of vortex method for fixed blades.

3.4 Nozzle optimization

Target functions in nozzle optimization are $r(x)$-type functions describing the shape of meridian nozzle section ($x$ – longitudinal coordinate, $r$ – nozzle radius), as well as inlet expansion ratio $\alpha$ (ratio between channel cross-sections at the nozzle inlet and in the propeller disk), outlet expansion ratio $\beta$ (ratio between channel cross-sections at the nozzle outlet and in the propeller disk) and relative nozzle extensions in certain cases.

For convenience of design, the surface of the nozzle can be split into three parts: external surface, internal surface upstream of propeller and internal surface downstream of propeller. Depending on operation conditions and purposes, it is possible to optimize both the entire surface of the nozzle and these parts severally. To optimize the external surface is to mitigate depressurizations and eliminate sharp pressure increases. The former condition improves cavitation performance of the nozzle, the latter one reduces the risk of boundary layer separation. To optimize internal nozzle surface upstream of propeller is to achieve constant pressure distribution throughout the nozzle surface. This optimization is sure to prevent cavitation and boundary layer separation upstream of the rotor. To optimize the area downstream of rotor is to achieve smooth pressure variation. All three parts of the nozzle are optimized simultaneously. In view of all these conditions, objective functional can be written as:

$$I = k_1 \int_{s_1} \left[ \frac{dC_p}{dx} \right]^2 \, dx + k_2 \cdot \frac{2 \cdot T}{\rho \pi R \bar{V}_r} + \begin{cases} \frac{dC_p}{dx}, & \frac{dC_p}{dx} < 0 \\ \frac{dC_p}{dx}, & \frac{dC_p}{dx} > 0 \\ k_3, & \frac{dC_p}{dx_i}, \frac{dC_p}{dx_i} > 0 \end{cases}$$  \hspace{1cm} (12)

where $C_p$ – pressure coefficient on the duct surfaces, $T$ – thrust of the unite, $k_1$, $k_2$ and $k_3$ are weight coefficients selected as per the given task.

Restricting conditions are as follows:

1) Specified flow rate must be preserved, i.e.
\[ 2\pi \int_{x_0}^{x} V_x r \, dr = Q^* \]  

where \( Q^* \) is specified flow rate in the nozzle, determined after calculation of propeller thrust and force at post-swirl stator, \( V_x \) – velocity in propeller disc;

2) The nozzle areas must be mated in a smooth manner, i.e.

\[
\begin{align*}
  r(x_i) &= r^*(x_i), \quad \frac{dr}{dx}(x_i) = \frac{dr^*}{dx}(x_i) \\
  r(x_f) &= r^*(x_f), \quad \frac{dr}{dx}(x_f) = \frac{dr^*}{dx}(x_f)
\end{align*}
\]  

(14)

where \( x_i \) and \( x_f \) – coordinates where the area to be optimized begins and ends; \( r^*(x) \) – radius of the initial nozzle,

3) Flow separations must be eliminated (this condition shall be formulated as per the parameters of the boundary layer on the nozzle surface).

Calculation of the flow around the nozzle is performed as per the method described in [14], along with the calculation of the boundary layer on its surface and the assessment of flow separation risk, see [18].

4 **DESIGN OF RSN PROPULSOR: CASE STUDY**

Prior to optimization of nozzled rotor, it was investigated how the configuration of this propulsor affects its parameters. This investigation was necessary because nozzled propeller and RSN propulsor, although looking alike, have different hydrodynamic properties. In particular, the location of optimal efficiency of the RSN propulsor was found not to coincide with the efficiency optimum of known nozzled propeller series. Furthermore, special analytical and experimental assessment of the optimum location were performed for the RSN propulsor, with consideration of all its elements. The knowledge of the optimum point ultimately allows making RSN propulsor more efficient, light and compact.

After the main parameters of the propulsor under design were selected (see Table 1), the optimization was performed as per the procedure described above. The initial geometry of the rotor and the nozzle was the one applied in OST nozzles, see [3]. The rectifying apparatus had constant width and thickness distribution. Rotor and rectifying apparatus both had zero rake and skew of blades. The design included determination of optimal distribution for pitch and curvature of rotor and post-swirl stator along the radius, determination of nozzle shape, as well as its inlet and outlet expansion ratios. The objective of design was to maximize efficiency and achieve the best cavitation performance in design conditions. For the general view of designed propulsor, see Fig. 1.

As design proceeded, optimal shape was found for all the elements. In particular, final geometry of the nozzle eliminates depressurization zone at its front edge in design conditions and simultaneously ensures smooth pressure growth towards the rear edge (see Fig. 2), which prevents flow separation. Fig. 2 comparing pressure distributions along the internal and external surfaces of the designed nozzle and nozzle D19a (if it is applied on this propulsor) clearly demonstrates optimization results. Reliability of the analytical assessments made in the design, was confirmed by CFD calculation results for the optimized geometry (see Fig. 2).
Table 1: Key parameters of RSN propulsor.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Operational conditions</td>
<td></td>
</tr>
<tr>
<td>Поступь $J$</td>
<td>1.065</td>
</tr>
<tr>
<td>Torque coefficient $K_Q^*$</td>
<td>0.065</td>
</tr>
<tr>
<td>Rotor</td>
<td></td>
</tr>
<tr>
<td>Blade area ratio $A_e/A_o$</td>
<td>0.55</td>
</tr>
<tr>
<td>Blade number $Z$</td>
<td>4</td>
</tr>
<tr>
<td>Nozzle</td>
<td></td>
</tr>
<tr>
<td>Relative elongation $l/D$</td>
<td>0.6</td>
</tr>
<tr>
<td>Location of rotor in the nozzle, $x_p/l$</td>
<td>0.4</td>
</tr>
<tr>
<td>Stator</td>
<td></td>
</tr>
<tr>
<td>Blade number</td>
<td>5</td>
</tr>
</tbody>
</table>

Distributed hydrodynamic parameters of the rotor were investigated in details as per the results of CFD calculation and are provided in Fig. 3 below. This Figure shows that pressure distribution throughout the rotor blade does not have any clear peaks, which confirms that the blades were designed correctly.

Figure 2: Distribution of pressure coefficient throughout the nozzle:
  a) internal surface, b) external surface.

Figure 3: Pressures on the rotor blade as fractions of the maximum depressurization:
  a) pressure side, b) suction side.
RSN propulsor design is an integrated process, i.e. all the elements of RSN propulsor have optimal parameters only if they operate jointly. Thus, according to the design calculation, the rotor-stator couple practically does not have any losses arising from wake vorticity. This is confirmed by CFD calculation data provided in Fig. 4. The Figure shows that although the vorticity downstream of the rotor is high and nonuniform, it is practically absent downstream of the stator.

![Figure 4](image)

**Figure 4**: Speeds distribution a) tangential nonuniformity on a few radii before the stator, b) averaged speeds along radius before and after the stator.

Additionally, Fig. 5 shows the field of tangential speed $V_\theta$ at the nozzle outlet. Diagram (a) shows that tangential speed at the nozzle outlet in design conditions is practically zero. The non-uniformity in the upper segment occurs because of the overpressure generated by the strut. Along with it, diagram (b) shows that in non-design conditions tangential speed is non-zero all over the section. Accordingly, in the latter case the stator does not fully use the energy of turbulent wake behind the rotor. Comparison of these two diagrams clearly shows that the stator must be selected on case-to-case basis, for given operational conditions of the rotor. Any attempt to use a universal stator will hinder propulsor efficiency.

![Figure 5](image)

**Figure 5**: Tangential speeds at the nozzle outlet as fractions of the apparent speed of the propulsor movement: a) design conditions; b) non-design conditions ($J = 0.75 \cdot J^*$).
Force parameters of the propulsion system versus its advance ratio are shown in Fig. 6. The same Figure compares them with CD calculation results for the full-scale propulsor. Achieved efficiency of the propulsor was remarkably high, 69%, as per the data of the design calculation, i.e. in model test conditions. For the full-scale propulsor, CFD calculation yielded the efficiency of ~75%, and this difference is due to scale effect.

![Figure 6: Performance curves of the designed propeller: predictions (as per the results of the verification calculation) vs CFD calculation results. Thrusts are shown as fractions of $\frac{\rho n^2 D^4}{4}$, torque is shown as fraction of $\frac{\rho n^2 D^5}{5}$.](image)

Deviation between CFD calculation data and verification calculation results at bollard pull ($J=0$) is due to flow separation on stator blades. It occurs because the stator is designed to high-speed operation, so at bollard pull are at a large attack angle with respect to the flow. It is noteworthy that no flow separation will occur if the propulsor is designed specifically for bollard-pull operation. But in this case, the propulsor, albeit good at bollard pull, will not have required performance at full speed.

5 EXPERIMENTAL VALIDATION AND OPERATIONAL APPLICATIONS OF RSN PROPULSORS

This optimization method for RSN propulsors, successfully evaluated in the case study described above, was applied to develop propulsors of AZIPOD XL family to be installed on future ABB OY ships. This family includes various sizes of propulsors with different motor power. Unlike previous azimuthal thrusters with nozzled propellers, mainly intended to operate at low speeds, RSN propulsors of this family are meant to be main propulsors of commercial ships running at the speeds of over 20 knots. These propulsors have high full-scale efficiency, ~75%, as per preliminary estimates, including those confirmed by CFD calculations.
The model tests of this propulsor family performed by KSRC confirmed accuracy and efficiency of the design calculations, as well as high performance of these propulsors. Fig. 7 compares design assessment of their performance parameters versus model test results.

6 CONCLUSIONS

- A design calculation method for RSN propulsor has been developed. This method is based on a set of analytical optimization techniques for all the elements of the propulsor, thus being considerably different from conventional methods of propulsor design;
- Optimization approach to design of RSN propulsors allowed identification their advantages, as well their utilization to the full. To achieve high parameters of RSN propulsor, the entire totality of its elements (rotor, stator, nozzle, etc.) must be optimized in an integrated manner;
- CFD calculations and model test data confirmed the efficiency of this method;
- This optimization method allowed the development of AZIPOD XL family of new highly efficient propulsors.

REFERENCES


[21] Model manufacture, propeller models terminology and nomenclature for propeller geometry. *ITTC procedure 7.5-01-02-01*.
OPERATION OF THRUSTERS IN ARCTIC WATERS
ARCTIC THRUSTER ECOSYSTEM

MARINE 2017

PROF. DR. BERTHOLD SCHLECHT*, DIPL.-ING. FREDERIK MIETH*,
DIPL.-ING. MANUEL KOSTIAL*, DR.-ING. THOMAS ROSENLÖCHER*,
DR.-ING. STEFAN SCHUMANN*

* Technische Universität Dresden (TUD)
Institut für Maschinenelemente und Maschinenkonstruktion IMM
Münchner Platz 3, 01069 Dresden, Germany
E-mail: Frederik.Mieth@tu-dresden.de, web page: https://tu-dresden.de/imm

Key words: Numerical Methods, artic condition, optimization of bevel gears efficiency

Abstract. Due to collisions between ice and propeller the drive train of ice breakers in artic conditions is a highly stressed system. Aim of the ArTEco ‘Arctic Thruster Ecosystem’ project is to increase the reliability of vessels when overloads and torsional vibrations occurs.

To achieve this aim different load scenarios will be analysed on a test rig located in Tuusula (Finland). Here the WST14 azimuth thruster, which is equipped with measuring instruments, is operated by the VTT and Wärtsilä. To investigate the behaviour of the test rig and the thruster different simulation models are created. These multibody system simulation (MBS) and finite element models (FE) are required to understand the behaviour of the thruster and investigate improvement strategies. Targets of these investigations are the dynamic behaviour during ice contact and the optimization of bevel gears in prospect of safety and efficiency. Therefor estimated propeller loads that occur during ice contact of the gear box housing or loads occur by hitting the propeller blades are used.

The simulation models of the thruster regards the flexible structure of the housing and shafts. Using this information the comparison considers natural eigenfrequency correlates to the test rig in Tuusula and the misalignment of the bevel gear can be investigated, validated and an optimisation achieved. For further analysis a simulation model is assembled by TU Dresden and verified by several time based data sets and modal analysis of the test rig. That way overloads and high dynamic loads which can’t be applied on the test rig are evaluable. Besides the dynamic analysis a progress in design phase for bevel gear stages is done. This is achieved by using complex FE models including the elastic bevel gear contact, bearing stiffness, clearances and the support of the flexible housing. The complex load and temperature condition lead to different displacements of the gears. Using simulation based displacement data and the software BECAL [1] a precise contact pattern can be investigated to determine safety factors, damage sum and efficiency.

To investigate the possible efficiency improvements of a bevel gear a design process is described. Therefor a variation of macro- and micro geometry were made. Point of interest is to shift the theoretical pitch cone relative to the contact pattern, to reduce the local sliding speed. The combination of profile shifting \( x_{hm1} \), pressure angle \( \alpha_{db} \) and profile crowning \( c_{pz} \) modification leads to a significant efficiency improvement.
1 INTRODUCTION

The main objective of the ArTEco project is an improvement of ship drive trains in arctic conditions, with the target to identify critical areas in ship propulsion design and improve products for this industry. The thruster test facility established in Finland at the end of 2013 achieved significant progress in conceptual development. Test facility allows measurements in a way that is not possible when a unit is in operation. These facts led Wärtsilä to establish the propulsion test facility, the Wärtsilä Propulsion Test Centre (WPTC) in the Helsinki area together with the VTT. A 2 MW thruster can be run with full power at the test rig in Tuusula and the operational loads can be applied to the thruster by 11 hydraulic cylinders.

The facility offers a so far unique possibility to make technology progress suitable for future marine thrusters. The test rig allows the measurement of bearing loads, gear displacements and shaft torque. Furthermore, oil temperatures, shaft displacements and vibrations can be monitored. As shown in Figure 1, the thruster housing is mounted on the structure of the test rig and the drivetrain is connected to an electrical motor and generator to apply torque and speed to the drivetrain. Six hydraulic cylinders can be applied on loading unit at the end of the propeller shaft, to apply thrust and side forces and five on the thruster housing. The cardan shaft, which connects the propeller shaft with the gearbox input shaft. With an electric connection between generator and motor, it is possible to feed only the power losses to the system and use the generator’s energy to run the motor. In Figure 1, the load introduction points are marked by red arrows.

![Simulation model, the red arrows indicate the load introduction points](image)

The institute of machine elements at TU Dresden develops the software BECAL (BEvel gear CALculation) which is an analysis tool for bevel gear design and safety evaluation. It was developed by the Institute of Machine Elements and Machine Design in corporation with the Institute of Geometry of the TU Dresden on behalf of the Drive Technology Research Association (FVA) [9]. The dynamic drive train behavior gets investigated using MBS methods since 2001. With these expertise the investigation of large drive trains, typical for ships, roller mills, compressors, fans, shearsers, cranes and wind turbines represent one of the main focuses of research in recent years [3], [4], [5].
2 TEST RIG AND SIMULATION MODELS

The motor and generator are the system boundaries of the drive train, with the thruster and the deceleration gearbox placed in between. The flexible structure of the thruster housing including the plate structure of the test rig is part of the simulation model as well. The connection to the floor is not considered, due to its high stiffness compared to the other components. The displacement behavior of the Thruster in the MBS Model is validated with an FE model including elastic housing, shafts, bearings, and gear contact (Figure 2).

![Figure 2: FE model (left) and MBS model (right)](image)

Before using the possibilities of the different simulation models the test rig gives the opportunity for validation. The validation is done by using the measurement results from modal analysis and time domain torque and displacement signals. The dynamic behavior of the test rig is determined by acceleration sensors during a modal analysis. The MBS model is using information about inertia and stiffness of the drivetrain components to calculate the natural frequencies of the test rig. The natural frequencies of the torsional system, the 6-DOF-MBS model and measurement results are compared in Figure 3. The comparison of simulated and measured natural frequencies of the thruster mode shapes in longitudinal, transversal and vertical direction shows a good agreement.

With the validated model, the system behavior can be predicted and tests can be rerun to analyze bearing loads and bevel gear stress with BECAL [1]. The FE model is already in use to calculate the bevel gear misalignment for the ongoing design procedure. In the current design process overload scenarios that represent the arctic loading spectrum are calculated and safety parameters are analyzed before they are applied to the test rig.

![Figure 3: Comparison of natural frequencies of simulation model and measurement](image)
3 BEVEL GEAR DEVELOPMENT

The developed FE and MBS models are used to design a new stage of bevel gears for the considered thruster. Figure 4 illustrates the workflow for the design process using the FE model and the software BECAL.

![Figure 4: Design studies for bevel gear geometry](image)

The occurring load vector $F$ is a crucial input parameter which correspond to the propeller loads during vessel lifetime. Investigation of arctic conditions leads to determine the load spectrum of the drive train. Resulting from the flexibility of the thruster housing, shafts and bearings, the reaction of the gear is a misalignment. This leads the contact pattern to shift from an ideal position to heel or toe and root or tip. Adjustment of the micro geometry of the tooth flank has to be done. Especially lead crowning, profile crowning and angular modifications are used to ensure a centered contact pattern. With the results of the displacement analysis, BECAL allows conclusions regarding the safety factors of the gears [1]. Optimization target in this workflow is mainly to lower the Hertzian contact and root stress. For the nominal load the second developmental stage of the Tuusula bevel gear results in 11% reduction of the Hertzian stress (Figure 5).

![Figure 5: Reducing the Hertzian contact stress of the thruster bevel gear for nominal load](image)

- a) developmental stage one, b) developmental stage two
4 ARCTIC CONDITIONS - ICE LOADS

One target of the ongoing investigation is to determine the load spectrum of arctic conditions. In cooperation with the VTT, a MATLAB tool for ice load simulation is created. These impact tools can be connected to the MBS model to investigate the load spectrum of arctic conditions. Ice peak loads correspond to ventilation, radial ice contact and ice crushing, which determines the propeller forces and probability according to [2], [6], [7]. These load spectrum can be applied to the Tuusula test rig via several hydraulic cylinders or ice blocks that mechanically hit the thruster housing. The IAS ice class represents the heaviest ice loads in the Finnish-Swedish ice class rules [6]. Figure 5 presents the ice load probability and amplitude for an open propeller (IAS) with the dimension of the investigated thruster in Tuusula. To describe the ice load probability a Weibull distribution is used [7]. The 100 % load amplitude in figure 6 represents the nominal load of the thruster.

![Figure 6: the ice load probability and amplitude](image)

BECAL gives the opportunity to calculate a damage accumulation for every local point on the flank [8]. The ice load distribution is used to calculate a Miner elemental accumulation for the bevel gear design stages. To validate the new stage of development the load distribution is calculated for a thruster operating in the IAS ice class for a lifetime of 5 years with nominal load. With the new variant the damage accumulation of the pinion flank can be reduced by 55% (Figure 7).

![Figure 7: Normalized damage accumulation for the pinion flank a) stage one b) stage two](image)
5 EFFICIENCY INVESTIGATION

After enhancing the load capacity another point of interest is to improve the efficiency. With BECAL a local gear loss can be calculated. For each point on the flank \((Y)\) the local gear loss \(P_{VZP}\) is calculated by the formula (1). It is related to the local normal load \(F_n(Y)\), the local sliding speed \(v_g(Y)\) and the local friction coefficient \(\mu(Y)\). The sum gear loss \(\bar{P}_{VZP}\) is calculated for the overall mesh with formula (2).

\[
P_{VZP} = \mu(Y) \cdot F_n(Y) \cdot v_g(Y) \quad (1)
\]

\[
\bar{P}_{VZP} = \frac{\sum \mu(Y) \cdot F_n(Y) \cdot v_g(Y)}{m} \quad (2)
\]

To investigate the possible efficiency improvements a test program will be done at the local bevel gear test bench located in Dresden. Therefore a variation of macro- and micro geometry of the test bench design where made. Point of interest is to shift the theoretical pitch cone relative to contact pattern, to reduce the local sliding speed. The aim is a pitch cone near to the middle of the contact pattern. Variation parameters are profile shifting \(x_{hm1}\), profile crowning \(c_{pz}\) and a profile angle modification \(H_{az}\) (Figure 8).

![Figure 8: Variation of a) profile shifting \(x_{hm1}\) b) profile crowning \(c_{pz}\) c) profile angle modification \(H_{az}\)](image)

To investigate these effect four different bevel gear designs are made. The contact pattern of these efficiency variants are presented in Figure 9. In the first step the profile shifting \(x_{hm1}\) is reduced from 0.5 (Variant A) to 0.3 (Variant B). Furthermore the profile crowning is increased from 50 to 150 \(\mu m\) (Variant C) and in the last step the pressure angle \(\alpha_{dd}\) is modified from 20° to 22° (Variant D). The combination of profile shifting \(x_{hm1}\), pressure angle \(\alpha_{dd}\) and profile crowning \(c_{pz}\) modification leads to a significant efficiency improvement. The bevel gear losses are reduced by 34%. These results will be validated at the local test bench located in Dresden.

![Figure 9: Hertzian stress for the macro - and micro - geometry variants](image)

\[
\eta_a = 98.90\% \quad \eta_b = 99.02\% \quad \eta_c = 99.22\% \quad \eta_d = 99.28\%
\]
6 CONCLUSION

The test rig established by the VTT and Wärtsilä gives all partners a better understanding of the complex behavior of azimuth thrusters. It provides the opportunity to analyze the systems behavior, which cannot be done during operation of a vessel. Several overload tests are executed to describe the drive train behavior under the load spectrum of arctic conditions. Further tests to detect the effect of biodegradable oils on gear fatigue and efficiency are planned. The interaction of test rigs and simulation models deliver a huge benefit for the participants of the Arctic Thruster Ecosystem project. Due to the time- and cost-efficient design studies a big step in improving the bevel gear design for thruster applications is already made and currently tested in Tuusula.

The Program BECAL is a Program of the Drive Technology Research Association (FVA) (http://fva-net.de/). It was developed by the Institute of Machine Elements and Machine Design in cooperation with the Institute of Geometry of the TU Dresden on behalf of the FVA.

This work was carried out under the Maritime Technologies II Research program in the framework of the ERA-NET MARTEC II project CA 266111 ArTEco ‘Arctic Thruster Ecosystem’.

REFERENCES

PARAMETRIC SEARCH AND OPTIMISATION OF FAST DISPLACEMENT HULL FORMS USING RANS SIMULATIONS OF FULL-SCALE FLOW

MARINE 2017

ERIC BRETSCHER*, STUART E. NORRISa, ANDREW J. MASONb, GREGOR J. MACFARLANEC AND JAMES P. DENIEd

* Department of Engineering Science,
The University of Auckland,
Auckland 1010, New Zealand
E-mail: e.bretscher@auckland.ac.nz, a.mason@auckland.ac.nz

a Department of Mechanical Engineering,
The University of Auckland,
Auckland 1010, New Zealand
E-mail: s.norris@auckland.ac.nz

b National Centre for Maritime Engineering & Hydrodynamics,
Australian Maritime College, University of Tasmania
Launceston, Tasmania 7250, Australia
E-mail: gregorm@amc.edu.au

c Department of Mathematics,
Macquarie University, NSW 2109, Australia
E-mail: jim.denier@mq.edu.au

Key words: RANS simulation, parametric modelling, shape optimisation.

Abstract. A methodology to derive parametric hull design candidates with a specified displacement and initial stability is introduced. A gradient-free search and optimisation algorithm coupled to a RANS CFD solver is then used to identify efficient pure-displacement hull shapes with minimal hydrodynamic resistance operating in the transition speed region without relying on dynamic lift.

1 INTRODUCTION

Fast pure-displacement monohull shapes have the potential to offer higher average operating speeds at much lower fuel consumption than planing or semi-displacement craft. Such hulls would be valuable in sectors as diverse as coastal fishing and leisure applications.

Historical research into fast displacement hull forms, as compiled and presented by Oossanen [1], identified that residuary resistance in the upper speed range is primarily a function of the length-to-displacement ratio. While a high length-to-displacement ratio appears to be a necessary condition, it is not in itself sufficient to achieve a low resistance. This paper shows that considerable potential exists to design small craft hull shapes, using viscous flow simulations coupled to an optimisation algorithm, which provide more
2 economical performance than is commonly found today.

2 PARAMETRIC HULL MODEL

2.1 Formulation

We created a fully parametric numerical hull model based on a nested parameterisation scheme in the modelling software CAESES V4.1.2 [2]. A hull cross-section model was first constructed using quadratic B-spline segments with 7 control points determined by 10 free variables. A broad diversity of hull cross-sections was able to be modelled, as illustrated in Figure 1. The tri-dimensional hull shape was then specified using control curves that defined values for the cross-section variables along the length of the hull. A bounded 41-dimensional master input parameter vector \( \vec{\rho} \in \mathbb{R}^{41} \) \( , \vec{l} \leq \vec{\rho} \leq \vec{u} \), allowed manipulating these control curves.

![Figure 1: Examples of achievable parametric half-hull cross-sections.](image)

This approach naturally supports the production of fair surfaces through simple continuity and differentiability conditions on the master parameter control curves. Unlike a deformation-based approach, the model is also able to evolve beyond its initial shape, while specific features can still be enforced through the bounds of the input vector as well as the intrinsic definition of the cross-section control curves.

The hull geometries were produced in a Cartesian coordinate system \((x, y, z)\) with its origin at the forward design waterline, where \(x\) is positive in the direction of the flow and the static waterplane is set at \(z = 0\). The modeller exported hydrostatic data for each hull as well as the geometries, most importantly the longitudinal position of the centre of gravity \(x_{CG}\), which is required for the computation of hull resistance in a free-to-trim situation.

2.2 Design constraint mapping

We eventually aim to solve an optimisation problem to minimise resistance while achieving specified initial stability and displacement targets. We handle these two constraints using a novel strategy to map the designs into an iso-displacement, iso-stable space. This ensures that the optimisation always compares hulls of equal capability, without resorting to normalisation of the hull resistance and the creation of Pareto fronts.

On the basis that, for a given scope of requirements and design intent, the elevation of the centre of gravity is not greatly impacted by the details of the hull shape itself, we specify the initial stability of the hull by setting a target value for the component \(BM\) [3], defined as:
where \( I_{WP} \) is the second moment of inertia of the waterplane area \( S_{WP} \) with regard to the hull centreline and \( V \) is the immersed volume of the hull:

\[
I_{WP} = \int_{S_{WP}} y^2 dS = 2 \int_{x=0}^{L_{WL}} \int_{y=0}^{Y(x)} y^2 dy dx = \frac{2}{3} \int Y(x)^3 \, dx
\]  

(2)

\[
V = \int_{V_{Hull,xco}} dV = \iiint_{V_{Hull,xco}} dx \, dy \, dz
\]  

(3)

where \( L_{WL} \) is the waterline length, and \( Y(x) \) is the waterline half-beam at position \( x \). Equations (2) and (3) show that the initial stability and displacement of any hull shape can be altered by scaling its beam by a factor \( \alpha_y \) and its depth (measured from the waterline plane \( z = 0 \)) by a factor \( \alpha_z \), giving:

\[
l'_{WP}(\alpha_y, \tilde{p}) = \alpha_y^3 l_{WP}(\tilde{p})
\]  

(4)

\[
V'(\alpha_y, \alpha_z, \tilde{p}) = \alpha_y \alpha_z V(\tilde{p})
\]  

(5)

The metacentric height \( BM' \) for the scaled design then becomes:

\[
BM'(\alpha_y, \alpha_z, \tilde{p}) = \frac{\alpha_y^2}{\alpha_z} l_{WP}(\tilde{p}) = \frac{\alpha_y^2}{\alpha_z} BM(\tilde{p})
\]  

(6)

If we assign the design target values \( BM' = BM_{Target} \) and \( V' = V_{Target} \), equations (5) and (6) form a trivial non-linear system that can be solved analytically for \( \alpha_y \) (7) and \( \alpha_z \) (8) in order to obtain a design based on parameters \( \tilde{p} \) satisfying equality constraints on displacement and metacentric height.

\[
\alpha_y = \sqrt[3]{\frac{V_{Target} \cdot BM_{Target}}{l_{WP}}}
\]  

(7)

\[
\alpha_z = \frac{1}{\alpha_y} \frac{V_{Target}}{V}
\]  

(8)

This transformation was programmatically automated within the parametric geometry modeller by performing a preliminary hydrostatic calculation followed by a subsequent scale transformation.

Lastly, it must be noted that hull beam and depth as such have no relevance within the set of design parameters and must be excluded, as they are uniquely determined by the final transformation.

3 RANS COMPUTATIONAL MODEL

The resistance of each candidate hull was calculated with Star-CCM+ V11.04.010R8, a code frequently used for hydrodynamic performance predictions of both displacement [4] and planing craft [5]. The problem was configured using the volume of fluid (VoF) formulation and high-resolution interface capture (HRIC) scheme to model the free surface. All simulations were conducted for calm water resistance in a free-to-heave and free-to-trim...
attitude. The HRIC scheme was configured to minimise numerical ventilation issues by preventing a switch to upwind differencing regardless of the local Courant number [6].

The computations employed a transient implicit formulation using the SIMPLE algorithm with a maximum of 8 iterations per time step. The flow was modelled as fully turbulent using the two-equation $k - \omega$ shear-stress transport model with an inlet turbulence intensity of 1%. Particular attention was placed on converging the solution down to very small residual errors.

3.1 Modelling motion

Hull motion was handled through the use of an overset mesh block encompassing the hull geometry. Care was taken to preserve compatible cell refinement sizes between the overset and background meshes, in accordance with the Star-CCM+ guidelines [7]. The width of the overset block was, following a sensitivity study, chosen as $0.3 \cdot \frac{L}{L_{WL}}$ in order to ensure that disturbances were not introduced into the formation of the divergent wave system; its length was minimised as much as possible to reduce rotation-induced mesh shear effects forward and astern of the hull.

3.2 Computational domain

The computations were carried out in a fluid domain simulating deep, open water for half a hull using a symmetry plane at $y = 0$. The origin of the computational domain was coincident with the origin point of the hull geometries at rest. The dimensions of the fluid domain are documented in Table 1.

<table>
<thead>
<tr>
<th>Table 1: Relative dimensions of the computational domain</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
</tr>
<tr>
<td>Width</td>
</tr>
<tr>
<td>Water depth</td>
</tr>
<tr>
<td>Inlet boundary distance</td>
</tr>
</tbody>
</table>

The length of the domain was established keeping in mind that the length of the transverse wave system increases with the square of the velocity and the insensitivity of the solution to the position of these boundaries was verified by performing systematic variations in distance between the stern and the outlet boundary.

3.3 Boundary conditions

The conditions applied at the domain boundaries varied between open water simulations and the simulation of tank experiments for validation (Table 2).

Inlet boundaries made use of the flat wave model available in Star-CCM+. The inlet flow velocity was ramped using a sine function from zero to the target simulation speed in 3 seconds and a momentum source based on the derivative of the inlet velocity profile was introduced into the fluid domain to deliver a matching acceleration. This approach produced stable solutions more quickly than an impulsive start, particularly in the upper speed range.
Table 2: Boundary type assignments

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Velocity inlet</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure outlet</td>
</tr>
<tr>
<td>Symmetry plane</td>
<td>Symmetry</td>
</tr>
<tr>
<td>Side</td>
<td>Free-slip wall</td>
</tr>
<tr>
<td>Bottom</td>
<td>Free-slip wall</td>
</tr>
<tr>
<td>Atmosphere</td>
<td>Velocity inlet</td>
</tr>
</tbody>
</table>

3.4 Mesh

A flow-aligned, trimmed, hexahedral mesh with local anisotropic refinement was used. The free-surface region was vertically refined uniformly throughout the domain and the wake region was further refined in the horizontal plane using block definitions aligned with the Kelvin wake angle [8]. Departing from common practices for ship hydrodynamics, significant additional refinement of the deep water body directly underneath and aft of the hull was introduced (Figure 2) in order to improve the behaviour of the solutions for simulations at Froude numbers $Fr \geq 0.6$.

![Figure 2: Mesh refinement regions.](image)

A combination of numerical wave damping and planar mesh coarsening was utilised to minimise reflections both at the inlet and outlet boundaries of the domain. The primary concern was avoiding reflection of the long transverse wavelengths produced in the upper speed range and modelling the wake accurately (Figure 3).

![Figure 3: Half-wake for a fast displacement hull candidate at $Fr = 0.6$ in simulated open water.](image)

The near-wall region was meshed using 8 inflation layers covering 60% of the estimated thickness of the boundary layer at the stern [9]. No meaningful changes in viscous or pressure forces were found when meshing the full thickness. The first cell height adopted was $y_0 = 0.54 \cdot 10^{-3} \cdot L_{WL}$, following a convergence analysis of the viscous forces, and yielded an
average non-dimensional value of $y^+ \approx 45$ at a Froude number $Fr = 0.6$, well into the logarithmic region. A geometric growth rate of 1.25 ensured a uniform size transition into the core mesh.

4 EXPERIMENTAL VALIDATION OF COMPUTATIONAL MODEL

4.1 Validation results against DSYHS hulls

We first validated our numerical flow model against the experimental results from selected models within the Delft Systematic Yacht Hull Series (DSYHS) [10]. The average magnitude of the error in the predicted resistance over the range $Fr = [0.3 ; 0.6]$ was 0.9% for hull SYSSER62, 1.6% for hull SYSSER35 and 1.8% for hull SYSSER48 (Figure 4).

![Figure 4: Plot of the experimental and calculated total resistance for selected Delft hulls.](image)

We observed deficits in predicted resistance above $Fr = 0.6$ for those hulls for which experimental data was available for such speeds. The bow of the Delft yacht hulls lifts out of the water as the dynamic trim increases with speed, leading to a strong pressure point with very high gradients where the blunt hull bottom parts the water. This unwanted phenomenon, which lies beyond the useful speed range of these designs, was imperfectly resolved by our mesh. Since it had no relevance for the type of hulls we were interested in, no further emphasis was placed on modelling it more accurately.

4.2 Validation results against test results at the AMC towing tank

We further tank-tested one of our design candidates, hull AMC 16-12, at the Australian Maritime College Towing Tank in Launceston, Australia, in compliance with the ITTC recommended procedure for calm water resistance tests [10].

Our hull AMC 16-12 was obtained using the optimisation process described in Section 6 and then built as a towing tank model. Unlike the Delft hulls, this design was suitable for operation in the transition speed region, where $Fr = [0.5, 1.0]$ approximately and a wave trough is present in the stern region. The characteristics of the model are detailed in Table 3.
Comparison between the numerical results and the experimental tests, summarised in Figure 6, Figure 7 and Figure 8, showed that the calculations over-predicted resistance at low speeds by up to 4.8% at $Fr = 0.25$. This discrepancy can be attributed to the fact that the computational mesh was not intended to be capable of resolving the very small and short waves experienced at low speeds, and to the fact that the very low magnitude of the forces being measured may have led to greater experimental uncertainty. In the range $Fr = [0.45, 0.85]$, resistance is consistently under-predicted by an average of -7.44% (Figure 6). We note that this linear error is observable at speeds where all the predictions for the Delft hulls were excellent.
Figure 6: Plots of the experimental and calculated total resistance versus Froude number for hull AMC-16-12.

Figure 7: Plots of the experimental and calculated trim versus Froude number for hull AMC-16-12.

Figure 8: Plots of the experimental and calculated sinkage versus Froude number for hull AMC-16-12.

The validity of the simulation was also assessed by comparing the computed dynamic trim (Figure 7) and sinkage (Figure 8) with the experimental data. Trim is consistently underpredicted by a small amount, a result also observed for all Delft hulls, but its behaviour is consistent. Sinkage predictions are excellent up to Fr = 0.6 and then diverge. Inspection of the time series collected from the tank tests shows that sinkage never reached a steady state.
and was still decreasing by the time the higher Froude number runs ended; amongst other known instabilities, the initial acceleration of the towing carriage can result in the model creating a low amplitude wave, which then interferes with the local water level and would require a longer run duration to subside.

5 SOLUTION ACCURACY

An estimate of the solution accuracy was derived from the mesh and time-step convergence studies at the design speed $Fr = 0.6$ used throughout all optimisation runs.

The Star-CCM+ automated mesher uses a base size specification $h_{Base}$ and locally-defined refinement and coarsening specifications. This size can be normalised against the hull waterline length as $h^* = h_{Base}/L_{WL}$.

![Figure 9: Plot of the convergence of the total resistance coefficient versus mesh size.](image)

The time-step can be expressed non-dimensionally in terms of the Courant number $CFL$ calculated against this base size, which was kept constant throughout mesh size changes. A convergent behaviour was found for $CFL \leq 1.0$. The coarsest mesh needed to be run at $CFL = 0.75$ in order to remain within allowable mesh motion limits at the overset boundary.

Figure 9 presents a plot of the total resistance coefficient versus mesh size, from which we can see a stabilisation for $h^* \leq 0.025$ the residual range on the total resistance coefficient is 0.045%. We determined that changes exceeding 0.05% could be considered as meaningful and simulations were conservatively carried out with a mesh base size $h^* = 1.97\%$ at $CFL = 1.0$ in order to avoid numerical ventilation issues for some hull variants.

6 FULL-SCALE RESISTANCE PREDICTION

Due to the differences in the development of the boundary layer, full-scale flow predictions for hull AMC 16-12 were sufficiently different (Figure 10) from the model-scale results to warrant carrying out hull shape optimisation work either directly at full scale or using an equivalent formulation, rather than relying on an extrapolation method.

We employed the procedure described by Haase et al. [11] to seek computed full-scale resistance from the flow simulation work carried out at model-scale by artificially increasing the Reynolds number to its full-scale value by reducing the water kinematic viscosity value.
This solution of the full-scale flow problem at model-scale proved more robust numerically and faster to solve than the true full-scale flow problem. Additional robustness was found without impacting the results by adopting the $k - \varepsilon$ realisable turbulence model and a high-$y^+$ wall function formulation for these simulations.

![Figure 10: Comparison of the full-scale (top) and model-scale (bottom) computed waterline for hull AMC 16-12 at $Fr = 0.6$.](image)

While validation of the full-scale resistance prediction is impossible in the absence of experimental data, we compared true full-scale resistance computations to the model-scale values obtained at the same Reynolds number to verify that the result followed the cube of the scale factor as expected. This result was confirmed by Haase et al. for slender catamaran hull shapes and it remains applicable for our non-slender hull shapes.

7 DESIGN SEARCH AND OPTIMISATION

7.1 Algorithm

A variety of optimisation algorithms and strategies were considered for this problem. Due to the relatively long CFD computation times required to obtain each hull resistance result, around 1 hour using 32 Intel x64 cores, we sought an optimisation algorithm both effective in terms of total number of evaluations required and able to exploit parallelism. This weighed against genetic algorithms (parallel with good exploration capabilities, but marginally efficient) and sequential methods such as Nelder-Mead. Gradient-based methods, including advanced algorithms using underlying local surrogate models like sequential quadratic programming, proved computationally intensive and failed to converge. One common critical issue was a lack of tolerance to failed design evaluations.

We used Asynchronous Parallel Pattern Search (APPS) [12], which is a parallel, gradient-free method with reasonable domain exploration capabilities that was specifically intended for such expensive “black box” problems presenting high-dimensionality and a superposition of modelling errors and numerical noise. APPS implementations can be found within the packages Dakota [13] and Hopspack [14]; we retained the Dakota implementation as it offers a unified framework for accessing a broad range of evaluation and optimisation tools.

APPS operates by exploring neighbours in a coordinate-search pattern around the current best computed point. For a problem featuring $m$ degrees of freedom, each pattern specifies $2m$ problems to be solved. We ran the APPS algorithm as a Parallel Pattern Search (PPS) by enabling blocking synchronisation and waiting until each pattern had been fully evaluated before formulating new input vectors. This makes a less intensive use of parallelism, but the execution is deterministic and easier to track, interrupt and resume if required.
7.2 Process outline

Our optimisation process was implemented using the solution pipeline shown in Figure 11. The generic optimiser Dakota was coupled to the parametric 3D modeller CAESES and run on a desktop computer, where it generated the parameter sets for CAESES. The resulting 3D geometries and hydrostatic data for the hull design candidates were then uploaded to a HPC cluster to obtain the flow solution at the target design speed from the CFD code Star-CCM+. The predicted hull resistance values were then returned to Dakota.

![Search and optimisation loop](attachment:image)

It is important to note that failures can occur at times for some designs. These can result from issues in modelling, geometry pre-processing, meshing or solving the flow problem and the optimiser must be able to recover and continue.

7.3 Targets and constraints

As the displacement and stability targets were met using the methodology presented in Section 2.2, only simple bounds were imposed on the design parameters to ensure that the natural validity limits of the geometric model were not exceeded. Some of these limits were also set in order to mandate specific design features and constrain the search within a desired envelope.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Full scale</th>
<th>Model scale</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Overall length</td>
<td>20.0</td>
<td>1.717</td>
<td>m</td>
</tr>
<tr>
<td>Displacement</td>
<td>22300</td>
<td>14.133</td>
<td>kg</td>
</tr>
<tr>
<td>BM</td>
<td>3.49</td>
<td>0.300</td>
<td>m</td>
</tr>
</tbody>
</table>

The target values for the hull displacement and initial stability were obtained from a fast 20-metre aluminium hard-chine lobster fishing boat (Figure 12) originating from Western Australia and surveyed in 2010. These are presented in Table 4 and reflect typical fast modern workboats of moderate displacement and low hull deadrise, where considerable initial stability, limited rolling motion and a lot of aft deck space are sought.
7.4 Initial hull shape and parameters

The initial hull shape was manually designed, drawing on hydrodynamic principles, with the objective of minimising any dynamic lift and run at low trim angles, thus relying primarily on hydrostatic pressure forces to support the hull at speed. The parametric model was then configured to closely approximate this shape and the corresponding parameter vector \( \mathbf{p}_0 \in \mathbb{R}^{41} \) was extracted.

This initial vector was first used in a centred parameter study with a total of 5 sampling points in each dimension. The dimensionality of the problem was then reduced by discarding variables that influenced the hull resistance coefficient by less than 1%. The resulting set was further compressed to 18 parameters by retaining those that showed a potential for improvement greater or equal to 0.2%, leading to an optimisation input vector \( \mathbf{p}' \in \mathbb{R}^{18} \). Reducing the parameter space is by no means a necessity as long as sufficient time or parallel computing capacity is available.

7.5 Hull shape optimisation

Figure 13 shows the solution history from two cascaded optimisation runs, where the second run was started using the best hull design found in the first run. We typically computed up to 36 solutions concurrently approximately every hour, using up to 1152 CPU cores. A total of 652 design variants were produced and 23 successively improved models were found by the algorithm. Thirty-six designs, or 5.5%, were not evaluated due to failures along the modelling and solution pipeline. The search yielded a 15.6% reduction in total resistance over the original hull while preserving the stability component \( BM \) and displacement for all designs.

7.6 Convergence considerations

Convergence of the APPS optimisation algorithm to a local minimum was proved by Kolda [12], where the stopping criterion is based on the contraction of the search pattern size below a minimum relative step size in all search directions. If the step size used in producing the search patterns around the best current solution becomes too small to produce meaningful changes in the objective function, the algorithm can terminate after failing to expand the pattern size again due to the impact of numerical noise and modelling errors. As shown in Figure 13, where a minimum step size of 5% of the range of each variable was used, we obtained further improvements by restarting the optimisation from the best point obtained after the first search ended: restarting has the effect of maximizing the step size again.

As APPS focuses on the best current solution, lower ranking, but different and potentially
attractive designs can be found without getting explored further during the search. Using these solutions as seeds for additional optimisation runs afterwards can lead to interesting results and lower minimums.

![Solution history for two cascaded Parallel Pattern Search runs.](image)

**Figure 13** - Solution history for two cascaded Parallel Pattern Search runs.

8 CONCLUSIONS

We have created an interesting and original low-resistance hull shape of pre-determined length, displacement and stability using optimisation based on full viscous flow simulations. The approach we introduced to meet displacement and stability targets is particularly suited to the design of small to mid-size craft, where form stability dominates and the height of the centre of gravity doesn’t normally represent a significant design challenge.

The work presented in this paper led to hulls offering approximately 50% less resistance throughout the displacement and transition speed ranges than the hard-chine lobster fishing vessel used as the source for representative initial stability and displacement figures.

9 ACKNOWLEDGMENTS

The authors want to thank Friendship Systems AG and CD-Adapco, which both made their software available for this work, as well as the Australian Maritime College at the University of Tasmania for providing access to their towing tank and supporting the experiments.

We also acknowledge the contribution of the NeSI high-performance computing facilities to the results of this research. New Zealand’s national facilities are provided by the New Zealand e-Science Infrastructure and funded jointly by NeSI’s collaborator institutions and through the Ministry of Business, Innovation & Employment’s Research Infrastructure programme.

REFERENCES

[10] (ITTC), I.T.T.C., Recommended Procedure 7.5-02-02-01, Resistance Test, 2011.
Propulsion Performance Optimization of “Neighbour Duct” by CFD
Kenta Katayama*, Yoshihisa Okada*, Yasuo Ichinose† and Ryohei Fukasawa‡

* Propeller design department, NAKASHIMA PROPELLER Co.,Ltd.
688-1, Joto-Kitagata, Higashi-ku, Okayama 709-0625, Japan
e-mail: ken-katayama@nakashima.co.jp, yoshihisa@nakashima.co.jp
web page: http://www.nakashima.co.jp

‡ Fluids Engineering and Hull Design Department, National Maritime Research Institute
6-38-1, Shinkawa, Mitaka-City, Tokyo 181-0004, Japan
e-mail: ichinose@nmri.go.jp, fukasawa@nmri.go.jp
web page: http://www.nmri.go.jp/index_e.html

Key words Energy-Saving Device, Duct, CFD, Optimization, EEDI.

Abstract. As one of measures against CO₂ reduction regulation by EEDI, energy-saving device (ESD) has been widely used. As one of ESD, the authors developed "Neighbour Duct" which was a vertical-long-oval stern duct.

Neighbour Duct generates thrust by harnessing flow along both sides of stern. By CFD, the geometric parameter of Neighbour Duct was optimized, and the principle of thrust generation was made clear.

In order to verify the result of CFD, a series of model test was carried out at National Maritime Research Institute (NMRI), the thrust deduction factors of both CFD and model test results were good agreement. As Estimation of performance of actual ship based on the model test, BHP was reduced 4.4% by Neighbour Duct. In addition, 1-w only decreased by 1%. Therefore it was found that CO₂ reduction effect would be obtained by Neighbour Duct without changing the propeller or propeller design.

1 INTRODUCTION

CFD (Computational Fluid Dynamics) has become large-scale year by year, and it has become possible to solve the flow field around the hull or the propeller accurately in the past few years. For this reason, CFD can be utilized to relatively easily develop energy-saving device.

Furthermore, as the demand for energy-saving has increased as a result of the increasing trend of international CO₂ reduction such as EEDI regulation. Efficient method by CFD to develop energy-saving hull or device has been required.

Under such circumstances, the authors developed a stern duct by utilizing CFD. In the energy-saving device which is set in front of the propellers influences on the propeller revolution speed because of the operation point of the propeller changes. Therefore it is necessary to alter the design of propeller if the affect is large. Moreover, since the behavior of the bilge vortex is difference between the model and the actual ship, it must be to change the geometry of the energy-saving device [1]. However it is difficult to change the geometry of the device between the model and actual ship without numerous actual ship measurement...
results.
In this study, thrust deduction coefficient 1-t was focused on and a duct which improved 1-t due to generate thrust was developed. The optimization method and CFD result of the duct was verified, the performance of the duct was evaluated.

2 DESIGN OF “NEIGHBOUR DUCT”

2.1 Vertical-long-oval duct

Generally, in order to be generated thrust by a stern duct, it is necessary proper angle of attack of a stern duct against an upward flow or a downward flow due to a bilge vortex[2]. On the other hand, “Neighbor Duct” obtains thrust by harnessing flow along sides of hull, the duct has a vertical-long-oval shape so as to be along sides of the hull as much as possible.

The shape of the duct was defined by several parameters, and optimization calculation using CFD was performed while changing the parameters.

2.2 Optimization by CFD

2.2.1 Particulars of the hull and propeller

In the optimization, 82,000 DWT Panamax bulk carrier developed by NMRI was used as the target ship. Principal particulars of the hull are shown in Table 1, and the propeller particulars are shown in Table 2. In this study, CFD was carried out on the model scale in order to verify by model test.

<table>
<thead>
<tr>
<th>Table 1: Principal particulars of hull</th>
</tr>
</thead>
<tbody>
<tr>
<td>NMRI 82Pana_Max Bulk Carrier</td>
</tr>
<tr>
<td>Principal Dimension</td>
</tr>
<tr>
<td>Length Between Perpendiculars</td>
</tr>
<tr>
<td>Length on Designed Load Water Line</td>
</tr>
<tr>
<td>Breadth</td>
</tr>
<tr>
<td>Depth</td>
</tr>
<tr>
<td>Design Draft</td>
</tr>
<tr>
<td>Block Coefficient</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 2: Principal particulars of propeller</th>
</tr>
</thead>
<tbody>
<tr>
<td>Principal Dimension</td>
</tr>
<tr>
<td>Diameter</td>
</tr>
<tr>
<td>Pitch Ratio</td>
</tr>
<tr>
<td>Boss Ratio</td>
</tr>
<tr>
<td>Expanded Area Ratio</td>
</tr>
<tr>
<td>Chord Length at 0.7R</td>
</tr>
<tr>
<td>Number of Blades</td>
</tr>
<tr>
<td>Shaft Center Line Height</td>
</tr>
</tbody>
</table>
2.2.2 Calculation condition

In this study, SC/Tetra which is a commercial code was used for CFD. The calculation was used the double-body model, and the turbulence model was SST k-ω. Fig.1(a) shows the calculation domain, and (b) the state of mesh around the stern.

In the calculation domain, a hull model is placed in a semi-cylindrical watershed, the dimensions of the calculation area are based on the Length of perpendicular (Lpp) of the hull model. The distance from the inlet to the bow equals Lpp, the radius of the cylinder equals Lpp, the distance from the stern to the outlet is twice of Lpp. The propeller thrust and torque were obtained on the propeller disk in Fig.1(b) by infinitely blade propeller theory.

![Figure 1](image)

The computational grids were mainly unstructured tetrahedral grids, and the prism grids were used in the boundary layer. The overset grid approach is used for the duct. The grids around the duct are subdivided in order to improve accuracy of the calculation.

The total number of cells was 32.6 million, and the dimensionless distance y+ from the wall surface of the first layer of the boundary layer was 1 or less.

The coordinates are the center point of the propeller disk as the origin, the x axis in the bow direction, the y axis in the port direction and the z axis in the water surface direction.

In the calculation of self-propulsive condition, calculation at 3 different revolution speeds of propeller was performed at design ship speed, and the self-propulsion factor at the self-propulsion point was obtained by interpolation.

The hull resistance was calculated without the duct. This resistance was used regardless with or without of the duct when calculate of the self-propulsion factor. Since the double-body model is applied to stabilize the calculation, the wave resistance is evaluated by the tank test results, which obtained from the resistance test conducting at the 400 m towing tank at NMRI.

2.2.3 Optimization

For optimization, self-propulsion factor was calculated for each duct which has difference shape parameter. The ratio of 1-t, 1-w with and without the duct are assumed to be △1-t, △1-w. The performance of the duct was evaluated of the two values, and other factors were not considered.
Fig. 2(a) shows the calculation results. The vertical and the horizontal axis represent $\triangle l_{-w}$ and $\triangle l_{-t}$ to plot the values obtained in each self-propulsion calculation on the chart. At the point painted red in Fig. 2(a), $\triangle l_{-t}$ is 2.6% and $\triangle l_{-w}$ is -2.9%, and the shape at this point is decided as the optimized duct. Fig. 2(b) shows the geometry of the optimized duct.

![Figure 2: (a) The optimization result, (b) the geometry of the optimized duct](image)

In the geometric parameters of the duct, having a large influence on the self-propulsion factor were size of the duct, opening angle of the duct and the side projection shape of the leading edge of the stern duct. The side projection shape of the leading edge has a large influence on $l_{-t}$. From those, it was found that the mutual position of the leading edge of the duct and the side of the hull is important.

### 3 PRINCIPLE OF THRUST GENERATION BY “NEIGHBOUR DUCT”

#### 3.1 Analysis flow field around the duct

The principle of thrust generation was analyzed in the optimized duct by CFD in detail.

As $D_p$ is the propeller diameter, and a plane is put on $Z = 0.23 ~ D_p$ as shown in fig. 1(b). Fig. 3 shows the streamlines and the pressure distribution on the inspection plane in the state with and without the duct. The streamline passes through a point on the inspection plane indicated by purple ‘x’ mark in fig.3.

As shown in the streamline and the pressure distribution on the plane, the flow along the hull sides flows to the duct with proper angle of attack. Therefore the lift is generated in the duct and thrust is generated. However, since the inside of the duct becomes negative pressure, drag occurs to the stern.
3.2 Pressure analysis on hull and stern duct surfaces

In order to evaluate the force exerted by the pressure in the state of with and without the duct, the normal vector of each surface was multiplied by pressure, and the x direction component P_x of them was obtained.

Fig.4(a) shows the distribution of P_x on the surface of the hull in the state of with and without the duct. A positive value represents thrust and a negative value represents resistance.

Comparing P_x distribution with and without the duct, P_x turns from positive to negative in the region where the surface of the hull is covered with the duct, which indicates that the resistance increased on the surface of the hull with the duct.

On the other hand, at the distribution of P_x on the surface of the duct shown in fig.4(b) shows a positive value as a whole, and in particular, it can be found that a large thrust is generated at the leading edge from the side portion to the upper portion inside the duct.

3.3 Change in thrust by stern duct

In order to compare the resistance and the thrust by the duct, the force F_x which working on the surface of the hull and the duct is obtained from integration P_x. Each F_x is shown in Table 3. However, due to the change the flow into the propeller by the duct, the propeller
wake flow changes and the resistance value of the rudder changes. The authors focused only on the relation between the hull and the duct, not including the rudder.

From Table 3, it can be found that the thrust generated by the duct is larger than the hull resistance. Therefore the thrust is increased in whole. Since trade-off between hull resistance and duct thrust is important for increasing thrust, it is clear that the mutual position of the duct and hull becomes important. This confirms the finding that the influence of the mutual position of the leading edge of the duct and the side of the hull obtained by the optimization calculation is large.

Also, the front edge of the duct side part generates a large thrust and it can be said that increasing the thrust generating part using the vertical-long-oval shape is effective for improving thrust.

<table>
<thead>
<tr>
<th>Region</th>
<th>Hull [N]</th>
<th>Duct [N]</th>
<th>Total [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fx : without duct</td>
<td>-14.36</td>
<td>-</td>
<td>-14.36</td>
</tr>
<tr>
<td>Fx : with duct</td>
<td>-14.58</td>
<td>0.63</td>
<td>-13.95</td>
</tr>
<tr>
<td>Difference</td>
<td>-0.22</td>
<td>0.63</td>
<td>0.41</td>
</tr>
</tbody>
</table>

4 MODEL TEST

4.1 Model

In order to verify the results obtained by the optimization calculation, a towing tank test was conducted in a 400 m water tank at NMRI. Regarding the resistance value used for the self-propulsion test, the resistance test value without stern duct was used as a reference regardless of the presence or absence of the duct was same as CFD. Fig.5 shows a photograph around the stern of the model used in the test. The model of the duct was made of resin and made with a 3D printer.
4.2 Comparison between model test and CFD

4.2.1 Self propulsion factor

Fig. 6 shows 1-t and 1-w obtained from CFD and model test at design speed. According to the model test result, \( \Delta 1-t \) and \( \Delta 1-w \) are 2.7% and -0.9%, respectively. Comparing CFD with the results of the model test, the absolute values of 1-t and 1-w are somewhat different, but the trend is consistent. Although \( \Delta 1-t \) is a reasonable calculation result, it proved to be overestimating about \( \Delta 1-w \).

4.2.2 Wake distribution

Fig. 7 shows the wake distribution obtained from the CFD and the model test. Comparing the CFD and the result of the model test without the duct, the position of the bilge vortex center and the state of the downwash by the vortex are in good agreement with each other. However at the upper middle part of the propeller disk, 1-w of the CFD result was lower.

Also, looking at the difference between the two due to the presence or absence of a stern duct, the change of wake by the stern duct is similar, but the degree of decrease of 1-w behind the stern duct is larger for CFD.

From these results, it can be said that this calculation method tends to evaluate excessively the deceleration degree of the fluid behind the object, such as a stern end and a stern duct. Therefore, the self-propulsion calculation by CFD seems to overestimate difference of 1-w due to the presence or absence of the duct.
5 THE PREDICTION OF PERFORMANCE OF THE ACTUAL SHIP

The performance of the actual ship was predicted based on the model test results. EHP was calculated by three-dimensional extrapolation method from the resistance test result without the duct. The wake coefficient correction \( 1-W_s \) with the duct was obtained from Yazaki chart and the following equation[3].

\[
1 - W_2 = \varepsilon_0 \times (1 - W_{T_0}) - \Delta W
\]

\[
\varepsilon_0 = \frac{(1 - W_2)}{(1 - W_T)}
\]

\[
\Delta W = (1 - W_{T_0}) - (1 - W_T)
\]

\( W_{T_0} \): Wake factor without a duct
\( W_T \): Wake factor with a duct
\( \varepsilon_0 \): Wake correction factor without a duct

Table 4: The prediction of performance of the actual ship

<table>
<thead>
<tr>
<th>Condition</th>
<th>Design Draft</th>
</tr>
</thead>
<tbody>
<tr>
<td>( V_s ) (Knot)</td>
<td>14.2</td>
</tr>
<tr>
<td>( F_0 )</td>
<td>0.1555</td>
</tr>
<tr>
<td>Duct</td>
<td>w/o duct</td>
</tr>
<tr>
<td>EHP (kW)</td>
<td>5,327</td>
</tr>
<tr>
<td>( \eta_R )</td>
<td>0.979</td>
</tr>
<tr>
<td>( 1-t )</td>
<td>0.811</td>
</tr>
<tr>
<td>( 1-W_T )</td>
<td>0.573</td>
</tr>
<tr>
<td>( 1-W_s )</td>
<td>0.627</td>
</tr>
<tr>
<td>( \eta_H )</td>
<td>1.294</td>
</tr>
<tr>
<td>( \eta_os )</td>
<td>0.524</td>
</tr>
<tr>
<td>( \eta_s )</td>
<td>0.663</td>
</tr>
<tr>
<td>BHP (kW)</td>
<td>8,112</td>
</tr>
<tr>
<td>( N_s ) (RPM)</td>
<td>114.7</td>
</tr>
</tbody>
</table>

The performance prediction of the actual ship estimation results are shown in Table 4. Effects on each self-propulsion factor by the duct on the actual ship were improved by 2.7\% for \( 1-t \), 1\% for \( 1-W_s \), 0.9\% for \( \eta_R \) and 0.2\% for \( \eta_0 \) respectively, and BHP is reduced 4.4\%. The change amount of \( 1-W_s \) is relatively small, therefore there is little influence on the propeller operating point, it can be said that it is not necessary to change the propeller design greatly.
6 CONCLUSION

In this study, we developed “Neighbour Duct”, for generating thrust by harnessing flow along sides of hull, and optimization of the geometric parameters of the duct were carried out by utilizing CFD.

Comparison the presence or absence of the duct, 1-t and 1-w were improved by 2.6% and 2.9% respectively in the result of CFD.

It was found that not only the size and opening angle of the duct but also the mutual position of the leading edge of the duct and the sides of the hull influences the duct performance.

In the model test, 1-t and 1-w were improved by 2.7% and 0.9% respectively by the duct. Comparison the model test and the CFD result, the result of 1-t is a reasonable, however 1-w seems to be overestimate in CFD because of the flow behind the object is excessively decelerated. In this study, although 1-t and 1-w were evaluated equally in optimization, it seems that an evaluation method weighting 1-t is more appropriate.

Analysis of the principle of thrust increase of the duct using the CFD result showed that by setting the duct having proper angle of attack regarding to the flow of the side portion of the hull, thrust was found to occur. Therefore, a vertical-long-oval duct which increases the thrust generating part on the side seems to be considered effective for efficiency improvement.

The performance of the actual ship was predicted based on the model test results and it was found that the duct reduces BHP of 4.4%. In addition, it is found that 1-w was only decreased 1%. Therefore

The wake flow distribution near the top changes by the duct, therefore it would be considered the influence on cavitation and surface force. In addition, it is necessary to accurately estimation method of the flow behind the duct in order to consider the influence of cavitation by CFD.

ACKNOWLEDGMENTS

In conducting this research, Dr. Yasutaka Kasahara and Dr. Yusuke Tahara of National Maritime Research Institute provided valuable guidance and advice. I would like to express my sincere gratitude.

REFERENCES

RATIONAL DESIGN OF FAST BOAT HULL

Nitai Drimer*, Or Neuberg*, Yahav Moshkovitch* and Roey Hakmon*

* Technion Israel Institute of Technology
  Faculty of Mechanical Engineering
  Technion City Haifa 32000, Israel
e-mail: nitaid@technion.ac.il

Key words: Planing Boat, Fast Boat, Rational Design, Slamming, Hydro Elasticity

Abstract. We present a new method for the rational design of the hull structure of planing boats, which considers hydro-elasticity, structural dynamics and nonlinearity of geometry and material. The method combines rules calculations with analytical expression and analysis of FSI to a practical design procedure. A design example demonstrates a saving of 20% of the bottom plate thickness, relative to design by rules, while keeping the rules allowable stress. Exceeding the rules allowable stresses enables further reduction of weight. We designed and constructed a full scale research boat and held sea trials, measuring strains at high sampling rate. Our research boat has two sides of different construction: the port side is designed by rules, while the starboard side is designed by our rational design method, with 20% thinner plates and double spacing between the longitudinal stiffeners. We present a verification of our design method: A comparison of stresses between design by rules, rational design, and measurements in the sea trials shows: For the heavy side (designed by rules), rules, rational, and trials show similar stresses, so both rules and rational are applicable for design; While for the light side (rational design), the rules dramatically over assess the stresses, while rational and trials show good agreement. We therefore expect this study to advance the design practice, to obtain more efficient boats.

1 INTRODUCTION

Typically, the dominant load for the design of planing hulls is slamming, while sailing fast at head seas. Classification rules, such as [1,2,3,4], assess the structure by statically applying an empirical design pressure to the structural members, represented by linear beam theory. However, the slamming is a violent Fluid Structure Interaction (FSI), where hydro-elasticity and dynamic structural responses are important [5]. For a rigid boat, the slamming pressure may be assessed by analytical solution, developed by Von Kerman [6] and extended by Wagner [7]. To consider hydro-elasticity, as well as nonlinear structure (geometry and material), the water entry problem is being solved by numerical methods, such as: Arbitrary Lagrangian–Eulerian (ALE) Finite Elements (FE) formulation, see for example [8], and Smoothed Particle Hydrodynamics (SPH), see for example [9]. Although hydro-elastic water entry is widely studied by researchers, it is yet rarely considered by boat designers. The design by rules of offshore fast boats typically results with robust hull. This research offers a design method [10] for the hull structure of planing vessels, which considers hydro-elasticity and nonlinear structural dynamics. Our method combines rules, theoretical solutions and numerical analysis to a practical design procedure and leads to more efficient design.
Section 2 presents the design procedure. Section 3 presents the design and construction of our research boat. Section 4 presents preliminary results of the sea trials and a comparison between design by rules, rational design by our method and measurements at the sea trials. Section 5 presents our conclusions and direction of continuance study.

2 RATIONAL DESIGN PROCEDURE

The design procedure is presented in [10]. We follow it here with an example, which is the design of our research boat.

2.1 Preliminary design of the hull by applicable classification rules

For the design of our research boat we adopted RINA rules [4]. Here we indicate the design parameters used for the Port side, which is designed by rules, while at the Starboard side the scantlings is reduced based on our rational design method. Our method considers structural design, while hull design includes other aspects as: functional architecture, hydrostatics, and hydrodynamics. Hence, we focus on the structural design; while only mention other aspects for the sake of completeness. The relevant steps of design by rules are:

(a) Specify a design speed, at a related sea-state: $24\text{knots}$ at significant wave height $1\text{m}$.

(b) Preliminary design the hull and specify the geometrical parameters required to assess scantlings by rules: waterline length $7.2\text{m}$, greatest molded breadth $2.5\text{m}$, deadrise-angle (measured from the water plane to the V bottom) at the center of gravity $20.3^\circ$.

(c) Assess weights and hydrostatics and specify mass parameters, required for the structural assessment by the rules: displacement $3000\text{kg}$, LCG (Longitudinal Center of Gravity) $2.8\text{m}$, draft $0.45\text{m}$ and running trim $4^\circ$.

(d) Apply rules to assess quasi static design pressure. The calculations by rules assess maximum stresses in the structural members (plates, longitudinal stiffeners, transverse frames) and determine the scantlings accordingly. The rules apply beam bending theory, where the critical load is typically the slamming pressure, which is uniformly distributed along each "beam" and is statically applied. Each "beam" may be a strip of the bottom plate between two stiffeners, or a longitudinal stiffener between two transverse frames or a transvers frame between the hull sides. The slamming pressure is proportional to the design vertical acceleration at the LCG:

$$a_{CG} = \frac{50 - \beta_{CG}}{3555C_B} \left( \frac{T}{16} + 0.75 \right) \frac{H_s}{T} + 0.084 \frac{B_w}{T} K_{FR} K_{HS},$$

where: $\beta_{CG}$ is the deadrise angle in degrees at LCG and should be taken at the range of $10^\circ$ and $30^\circ$; $\tau$ is the running trim angle in degrees and should be taken at least $4^\circ$; $H_s$ is the significant wave height in meters; $T$ is the fully loaded draft in meters, at rest in calm water; $B_w$ is the greatest moulded breadth measured on the waterline at draft $T$; $C_B = \frac{\Delta}{\rho L B_w T}$ is the block coefficient, related to the waterline length $L$ in fully loaded displacement $\Delta$, the density of the water $\rho$ in ton/m$^3$, 1.025 for sea water; $K_{FR}$ and $K_{HS}$ are coefficients defined by:
\[
K_{FR} = \left(\frac{V_x}{\sqrt{L}}\right)^2 ; \quad K_{HS} = 1 \quad \text{if} \quad \frac{\Delta}{(0.01L)^3} \geq 3500 \quad \text{and} \quad \frac{V}{\sqrt{L}} \geq 3 \\
K_{FR} = 0.8 + 1.6 \left(\frac{V_x}{\sqrt{L}}\right) ; \quad K_{HS} = \frac{H_S}{T} \quad \text{if} \quad \frac{\Delta}{(0.01L)^3} < 3500 \quad \text{or} \quad \frac{V}{\sqrt{L}} < 3
\] (2–2)

In (2–2) \( V \) is the maximum service speed and \( V_x \) is the actual boat speed, in knots.

Table 2-1 summarizes the parameters of our research boat, for which \( a_{CG} = 4.97 \) g.

<table>
<thead>
<tr>
<th>( V_x )</th>
<th>( H_S )</th>
<th>( \tau )</th>
<th>( \beta_{CG} )</th>
<th>( B_w )</th>
<th>( T )</th>
<th>( L )</th>
<th>( \Delta )</th>
<th>( C_B )</th>
<th>( \rho )</th>
</tr>
</thead>
<tbody>
<tr>
<td>24 Knot</td>
<td>1.0 m</td>
<td>40</td>
<td>20.3</td>
<td>2.5 m</td>
<td>0.45 m</td>
<td>7.2 m</td>
<td>3 ton</td>
<td>0.36</td>
<td>1.025 ton/m³</td>
</tr>
</tbody>
</table>

Table 2-1: Boat Design parameters

The maximum slamming pressure on the bottom of the hull is given by (2–3) in kPa:

\[
p_{sl} = 70 \frac{\Delta}{S_r} K_1 K_2 K_3 a_{CG}, \quad S_r = 0.7 \frac{\Delta}{T}
\] (2–3)

In (2–3): \( S_r \) is a reference area, in \( m^2 \), \( K_1 \) is a factor for longitudinal distribution, defined by equation (2–4), where \( x \) is the distance, in m, from the aft perpendicular to the load point.

\[
\begin{cases}
K_1 = 0.5 + \frac{x}{L}, & \frac{x}{L} < 0.5 \\
K_1 = 1, & 0.5 \leq \frac{x}{L} \leq 0.8 \\
K_1 = 3 - 2.5 \frac{x}{L}, & 0.8 < \frac{x}{L}
\end{cases}
\] (2–4)

\( K_2 \) is a factor related for the impact area, defined by (2-5) and must be greater than: 0.5 for plating, 0.45 for stiffeners and 0.35 for girders and floors.

\[
K_2 = 0.455 - 0.35 \frac{u^{0.75}}{u^{0.75}+1.7}, \quad u = 100 \frac{s}{S_r}
\] (2–5)

In (2–5) \( s \) is the area supported by the element: for plating it is the spacing between stiffeners multiplied by their span, without taking for the span more than 3 times the spacing. By (2–5) the factor \( K_2 \) exceeds 0.5 only if \( \frac{s}{S_r} < \frac{1}{70} \) and this rarely happens for typical spacing between stiffeners. Hence, for plating, the factor \( K_2 \) typically equals 0.5.

The factor \( K_3 \) accounts for the variation of deadrise along the hull, and is given by (2–6), where \( \beta \) is the deadrise angle in degrees at the calculated section.

\[
K_3 = \frac{70 - \beta}{70 - \beta_{CG}}
\] (2–6)

Table 2-2 summarizes the parameters in our example.

The slamming pressure obtained at the mid-ship of our boat is \( p_{sl} = 112 \) kPa.
Table 2-2 – Parameters related to the design pressure by rules

<table>
<thead>
<tr>
<th>$x/L$</th>
<th>$K_1$</th>
<th>$K_2$</th>
<th>$K_3$</th>
<th>$s$</th>
<th>$S_r$</th>
<th>$u$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>1.0</td>
<td>0.5</td>
<td>1.0</td>
<td>0.106 m$^2$</td>
<td>4.67 m$^2$</td>
<td>2.26</td>
</tr>
</tbody>
</table>

The slamming pressure is calculated at every $x/L$ between transverse frames (for the calculation of the maximum bending stress at the bottom plating and at the longitudinal stiffeners), as well as at every $x/L$ of a transverse frame (for the calculation of maximum bending stress at the transverse frame). According to the rules, the pressure $p_{sl}$ is applied as a uniformly distributed load to a beam, which represents the local structural member. The end conditions are typically clamped-clamped, as a continuous situation is assumed, where the adjacent spans are applied to similar pressure. To comply with the allowable bending stress 138 MPa, specified by [4] for the boat material Al5083, we obtained a required thickness of bottom plate 4.8 mm for a span between stiffeners of 210 mm (at the prismatic boat section from transom to about mid-ship). Toward bow the slamming pressure is increased by pitch ($K_t$), however decreased by the deeper V (deadrise), and the thickness is sufficient. Figure 2-1 presents the structure of our research boat.

Figure 2-1: Hull Structure, designed by rules

2.2 Theoretical assessment of quasi static pressure

Wagner [7] presented a theoretical solution of the slamming pressure, applied to rigid prismatic wedge entering incompressible water at a constant vertical velocity:

$$p - p_a = \rho V_z \left( \frac{c}{(c^2 - y^2)^{1/2}} \frac{dc}{dt} + \rho \frac{dV_z}{dt} (c^2 - y^2)^{1/2}, c(t) = \frac{\pi V_z t}{2 \tan \beta}, \right)$$

where: $\rho$ is the water density; $V_z$ is the vertical drop velocity, $y$ is a coordinate from the keel across the bottom, $c$ is the wetted half beam, $\beta$ is the deadrise angle and $t$ is the time from the wetting of the keel. The assumptions of rigid hull and incompressible fluid, result with infinite pressure for a hull of zero deadrise and at $y = \pm c$ for any deadrise. Faltinsen [5] suggested assessing the design pressure by a space-average along the loaded interval, from $y_i$ to $y_{i+1}$ (see Figure 2.2) of Wagner’s solution (2–7), assuming a constant vertical velocity. The
maximum space-averaged pressure, $p_{\text{av}}^{\text{max}}$, occurs when $c = y_{i+1}$. The integration gives a finite average slamming pressure:

$$p_{\text{av}}^{\text{max}} - p_{\infty} = 0.5\rho V_z^2 \frac{\pi}{\tan \beta \left( \frac{y_{i+1}}{y_{i+1} - y_i} \right)} \left( \frac{\pi}{2} - \sin^{-1} \left( \frac{y_i}{y_i + 1} \right) \right)$$  \hspace{1cm} (2–8)

2.3 Finding a corresponding drop velocity

The analytical pressure (2–8), as well as our hydro-elastic numerical simulations (presented in the following Item 2.4) are for a vertical drop of prismatic sections into the water (two dimensional problems). The design calculations by classification rules relate the slamming load at each location along the boat to the design service conditions: design speed at a design sea-state, defined by the significant wave height. Therefore, interpretation of theoretical or numerical results of water entry to the design of a boat requires correlation between the drop velocity and the design service conditions. We define that the vertical velocity of water entry (drop velocity), which corresponds to the design service conditions (speed and sea-state), is the velocity that causes the same slamming pressure: by the theoretical solution (2–8) for a rigid prismatic wedge entering the water at the corresponding drop velocity and by the rules for a boat sailing at the design service conditions (2–3). Both assessments, the theoretical and by the rules, are for a rigid hull (which means quasi static pressure) and are two dimensional (for a finite strip of the boat at specific $x$). Equating (2–3) and (2–8) we obtain:

$$V_z = \left[ \frac{140 \Delta}{S_y} K_1 K_2 K_3 a_{CG} \right]^{1/2} \left( \rho \frac{\pi}{\tan \beta} \frac{y_{i+1}}{y_{i+1} - y_i} \left( \frac{\pi}{2} - \sin^{-1} \left( \frac{y_i}{y_i + 1} \right) \right) \right)$$ \hspace{1cm} (2–9)

In our example, the corresponding drop velocity at the mid-ship, is $V_z = 4.0 \text{ m/s}$. As the slamming pressure depends on the location along the hull and on the deadrise angle ($\beta$), which is increased toward bow, several drop velocities are needed to be found, along the hull.

The concept of the corresponding drop velocity practically accounts for the random nature of the sea. The rules assess a design slamming pressure proportional to the design vertical
acceleration, which is related to the significant wave height. According to RINA rules [4], which we adopted, the design vertical acceleration corresponds to the average of the highest 1% of the random vertical accelerations. By the Rayleigh probability density function (PDF), which is universally used to represent the statistical distribution of wave heights, the average of the highest 1% of the waves is $1.67$ times the significant wave height.

2.4 Hydro-elastic simulations and their implementation to boats design

At Stage 2.3 we obtained for each transvers cross section along the boat, a corresponding drop velocity. A prismatic (two dimensional) rigid section of the hull, with a deadrise as the local deadrise of the boat, vertically entering the water at the corresponding drop velocity, will be applied to the same design pressure calculated by the adopted rules at the design sailing conditions. While the rules calculations apply linear beam theory and are restricted to a rigid hull, as they specify quasi-static pressure; the numerical simulations account for hydro-elastic interactions, dynamics, and nonlinear structural analysis. Thus we need to solve 2D problems of fluid structure interaction, where each section of the boat enters the water at the corresponding drop velocity. As a very fine time stepping is required for the solution of the water-structure interaction, there are no extra efforts in specifying non-linear material and non-linear geometry as well.

While applying the corresponding drop velocity to a deformable hull, we assume that the deformation, which may be important for the local FSI, does not affect the motion of the boat at sea; hence will not affect the correlation between the design service conditions (boat speed and wave height) and the vertical drop velocity, which was obtained for a rigid hull.

In the present study we apply the commercial code ABAQUS/CAE with Arbitrary Lagrangian–Eulerian (ALE) formulation for the fluid domain and Lagrangian formulation for the structure domain. The formulation of the conservation equations of momentum and mass for solving the water domain are presented for example by [8].

2.4.1 The model

The model (Figure 2-3) represents a strip of the bottom plate, under the assumptions:
- The problem is two dimensional ($x$ independent);
- No air entrapment in the water or air cushioning between the water and the boat;
- The problem is symmetrical about the $y$-$z$ plane, so only half section is modelled.

![Figure 2-3: Model representation of the boat section](image-url)
The ALE formulation in ABAQUS does not include a 2D option. The $x$ independency is obtained by using a single element along the boat, and applying Symmetry boundary conditions at the transverse ($y$-$z$) planes, aft and forward of the modeled strip.

The boat section is represented by an assembly of three parts: elastic bottom plates; rigid rods, which represent the supports of the bottom plate by the longitudinal stiffeners; and a rigid frame from above (see figure 2-3). The elastic bottom is modeled by four nodes shell elements ($S4R$ in ABAQUS). The rigid parts, stiffeners and frame are defined as "rigid bodies". The structure domain includes 1400 elements of size $2mm$. The constraints between the elastic bottom part and the rigid frame are fixed at the keel and at the chine. The inner stiffeners are pinned to the elastic bottom and to the rigid frame.

The fluid domain is modeled by linear Eulerian brick eight nodes elements ($EC3D8R$). The dimensions of the fluid domain are: width $1.6m$, depth $0.4m$ and thickness (along $x$) $2mm$ (element size). An additional grid of height $0.2m$ is included above the water line for the run-up. For the element size of $2\times2\times2mm$ we obtained 240,000 elements in the water domain.

### 2.4.2 Boat Material

An experimental strain-stress curve, transformed to "true" strain, presented by Figure 2-4, was used as the material constitutive law. Table 2-2 presents the boat material properties.

<table>
<thead>
<tr>
<th>Material</th>
<th>Aluminum 5083 H321</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's modulus $E$</td>
<td>52.9MPa</td>
</tr>
<tr>
<td>Poisson ratio $\nu$</td>
<td>0.3</td>
</tr>
<tr>
<td>Density $\rho$</td>
<td>2700kg.m$^{-3}$</td>
</tr>
</tbody>
</table>

**Table 2-2: Boat material properties**

### 2.4.3 Database for design

This section presents results of the simulations of water entry, over a wide range of parameters, which we consider sufficient for a practical range of design. The results map the stresses over transverse sections of the bottom plate, for different deadrise angles, plate thicknesses, spans between stiffeners and velocities of water entry. We carried out a systematic series of 225 simulations by programming a parametric script in Python (the script language used by ABAQUS). Table 2-3 presents the varying parameters.

Figure 2-5 presents an example of the pressure distribution along the structure-water interface, together with the structure deformation at two times during the water entry.
Variable | units | Cases |
--- | --- | --- |
Drop velocity – $V_z$ | m.s$^{-1}$ | (-3, -4, -5) |
Length between stiffeners – $L$ | mm | (210, 315, 420) |
Bottom Plate thickness – $d$ | mm | (3.2, 4.0, 4.8, 6.0, 8.0) |
Dead rise angle – $\beta$ | degree | (7, 14, 21, 28, 35) |

Table 2-3: Varying parameters of the simulated cases

Figure 2-5: Water surface, structure deformation and distribution of contact pressure at successive times of water entry, for a drop velocity $V_z = -4$ m.s$^{-1}$, $\beta = 21^\circ$ and $L = 315$ mm

At the first time presented, the jet location is at the first stiffener. As the second span is not loaded, the right boundary condition for the first span is more like a simple support and not a clamp, as assumed by the rules. At a later time, loading of the second span make the first span to deform similar to a clamped-clamped beam. Modeling the bottom plate as a continuous beam supported by the stiffeners is more representative than the rules calculations and does not require an assumption regarding the end conditions for each span.

As the pressure distribution varies dramatically during the water entry, it is not trivial to guess the time step at which the load effects are most critical. Hence, for each analysis a maximum strain envelope (maximum over time for each location) is stored and presented. As the simulation time is long and 225 cases were processed, we limited the duration of the solution to the water entry of the middle of the second span. Simulations for the whole process show that the maximum strain envelope at the first span is not varied while continuing the simulation. Figure 2-6 presents examples of the strain envelopes for the whole process of water entry.
Figure 2-6: Maximum strains envelop (over time) along three spans, for $V_z = -4 \text{ m/s}$, $\beta = 21^\circ$ and $d = 4 \text{ mm}$. Strains at the top and the bottom of the plate are shown by solid and dashed lines respectively.

Storing the maximum stresses over the time and location allow us to present a database of 225 simulations in a compact manner, usable for design. The results are gathered in three figures in [10], one for each velocity of water entry. Each figure presents 75 cases: 5 deadrise angles $\times$ 5 thicknesses $\times$ 3 spans. Angles of deadrise are distinguished by colors, spans by symbols and plate thicknesses by the horizontal axis. Calculation by linear beam theory, assuming theoretical uniformly distributed load and clamped-clamped end conditions (as typically assumed by boat design rules) are shown in continuous lines of different line-styles for different spans. These lines are trimmed at the yield stress, above which the linear theory is not valid. A verification of the simulations is presented by approaching the static-linear theory when the structure becomes rigid (thick plates and short spans).

As an example we may apply the hydro-elastic design charts to optimize the boat, for which we presented the design by rules. We use the design chart for the corresponding drop velocity that we obtained at mid ship, $V_z = 4.0 \text{ m/s}$, which is presented by Figure 2-7. A blue line presents the allowable stress of $138 \text{ MPa}$. It is shown on the figure that for a deadrise angle of $21^\circ$ the rules method results with a required plate thickness of $4.2\text{ mm}$, while only $3.4\text{ mm}$ is required by the rational approach; a reduction of $20\%$. The research side of our boat has plate thickness of $4\text{ mm}$ and the spacing between stiffeners is $420\text{ mm}$. By Figure 2-7 we obtain a maximum stress of $230\text{ MPa}$. For such high stresses, we will obtain more significant differences between rules calculations and rational analysis, as we present in section 4.

3 DESIGN AND CONSTRUCTION OF THE RESEARCH BOAT

We designed and constructed a full scale research boat and held sea trials. Our research boat, named Dganit, has two sides of different construction: the port side is designed by rules with bottom plates of $4.8\text{ mm}$ stiffened longitudinally at spaces $210\text{ mm}$, while the starboard side is designed by our rational design method, with $4.0\text{ mm}$ plates and double spacing, $420\text{ mm}$, between the stiffeners. Figure 3-1 presents the construction at B.M. Carmel.
The boat is equipped with an array of strain gages, located at points of high stresses (see Figure 4-3). The Stored Sampling rate is 2048Hz, which is processed by filtration of raw sampling at 40,000Hz.

4 PRELIMINARY RESULTS OF THE SEA TRIALS

Figure 4-1 presents our research boat at sea trials. Figure 4-2 presents typical strain records. As the sea is random, the comparison requires statistical processing of the measurements. Adopting the rules approach, we process the strain records of each leg, during which the sea state is assumed stationary, to find the mean of the highest 1% of stresses.
Figure 4-1: Research boat Dganit during sea trials

Figure 4-2: Example of Strains records at 26 knots Head seas Hs 1.1m

Figure 4-3 presents comparison of stresses between design by rules, our rational design, and trials measurements. The results are presented at spots of highest stresses, at the ends of spans. Due to the fillet welds, the strain gages are located about 20mm from these spots and the measurements are factored to represent the maximum stresses. The factors are obtained from the strain envelopes by analysis. The comparison clearly shows: For the heavy side (designed by rules), rules, rational, and trials show similar stresses; however, for the light side, the rules dramatically overestimate the stresses, while rational and trials are similar.

Figure 4-3: Comparison of stresses
5 CONCLUSIONS

This paper presents a rational design method for the structure of planing hulls, which considers hydro-elasticity and dynamic nonlinear structural response. A database of results of simulations of water entry, in a wide range of parameters, enable assessment by designers, without the need to setup and run FSI simulations.

Preliminary processing of measurements of sea trials of a full scale research boat, validate the method and show that for a light structure it is significantly more pragmatic than design by rules.

Currently, the method is extended to include a Fatigue Limit State (FLS) design procedure. We expect that this study will advance the design and production of more efficient boats.

ACKNOWLEDGEMENTS

This research was supported by the MEYMAD grant program of the state of Israel and by SELA Ltd. of Israel.

REFERENCES

SINKAGE, TRIM, DRAG OF A COMMON FREELY FLOATING MONOHULL SHIP

CHAO MA, YI ZHU, HUIYU WU, WEI LI, HUIPING FU AND FRANCIS NOBLESSE

State Key Laboratory of Ocean Engineering
Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration
School of Naval Architecture, Ocean & Civil Engineering
Shanghai Jiao Tong University, Shanghai, China
e-mail: chaoma1988dr@163.com

Key words: Sinkage, Trim, Drag, Monohull Ships, Practical Methods, Design

Abstract. A practical method — well suited for early ship design and hull form optimization — for estimating the sinkage, the trim and the drag of a freely-floating common monohull ship at moderate Froude numbers \( F \leq 0.45 \) is considered. The sinkage and the trim are realistically estimated via two alternative simple methods: an experimental approach based on an analysis of experimental measurements (involving no flow computations), and a numerical approach based on a practical linear potential-flow theory (the Neumann-Michell theory) that only requires simple flow computations for the hull surface \( \Sigma_{H0} \) of the ship at rest. The drag is also estimated in a simple way, based on the classical Froude decomposition into viscous and wave components: well-known semi empirical expressions for the friction drag, the viscous drag and the drag due to hull roughness are used, and the wave drag is evaluated via the Neumann-Michell theory. The drag is more sensitive to the hull position than the sinkage and the trim. Accordingly, it must be computed for a ‘dynamic’ ship hull surface \( \Sigma_{H_{st}} \) that accounts for sinkage and trim effects, although the hull surface \( \Sigma_{H_{a}} \) does not need to be very precise. In fact, the total drag computed for the hull surface \( \Sigma_{H_{st}} \) chosen as the hull surface \( \Sigma_{H_{a}} \) predicted by the numerical approach, or as the hull surface \( \Sigma_{a} \) predicted by the even simpler experimental approach, are nearly identical. Moreover, the drag of the hull surface \( \Sigma_{H_{st}} \) and the (nearly identical) drag of the hull surface \( \Sigma_{a} \) are significantly higher, and also in much better agreement with experimental measurements, than the drag of the hull surface \( \Sigma_{H_{0}} \) of the ship at rest at high Froude numbers.

1 INTRODUCTION

The drag experienced by a ship is a critical element of ship design. Accordingly, the prediction of the flow around a ship hull that advances at a constant speed along a straight path, in calm water of large depth and lateral extent, is a classical basic ship hydrodynamics problem that has been widely considered in a huge body of literature, e.g. [1].

The drag of a ship is well known to be influenced by several complicated flow features, including (i) viscous effects and the related flow separation that typically occurs at a ship stern,
notably a transom stern, (ii) nonlinear effects and the related wavebreaking at a ship bow, (iii) hull roughness effects for full-scale ships, and (iv) the influence of sinkage and trim for a freely floating ship. This study examines the influence of sinkage and trim on the drag of a common generic freely floating monohull ship (free to sink and trim) at \( F \equiv V/\sqrt{gL} \leq 0.45 \), where \( V \) and \( L \) denote the speed and the length of the ship, and \( g \) is the acceleration of gravity.

The pressure distribution around a ship hull surface \( \Sigma^H \) that advances at a constant speed \( V \) in calm water evidently differs from the hydrostatic pressure distribution around the wetted hull surface \( \Sigma^H_0 \) of the ship at rest, i.e. at zero speed \( V = 0 \). Consequently, the ship experiences a hydrodynamic lift and pitch moment, and a related vertical displacement and rotation of \( \Sigma^H_0 \) that are commonly called sinkage and trim, as well known and widely considered in the literature; e.g. [2-10].

As already noted, alternative methods for evaluating the sinkage and the trim, as well as the drag, experienced by a freely floating ship have been considered in the literature. In particular, the approach considered in [2-8] involve iterative flow computations for a sequence of hull positions, which is ill suited for routine practical applications to early ship design and hull form optimization, and are unnecessary at Froude numbers \( F \leq 0.45 \), as shown in [11]. Practical methods for estimating the sinkage, the trim and the drag of a ship, notably methods that do not require iterative flow computations for a sequence of hull positions, are useful (if not necessary) at early design stages and for hull form optimization.

[11] considers two simple approaches — an ‘experimental approach’ and a ‘numerical approach’ — for estimating the sinkage and the trim of a typical freely floating monohull ship that advances in deep water at a Froude number \( F \leq 0.45 \).

The experimental approach is based on an analysis of experimental measurements reported in the literature for 22 models of monohull ships. This analysis of experimental data yields particularly simple approximate analytical relations that explicitly predict the sinkage and the trim experienced by a monohull ship — without flow computations — in terms of the Froude number \( F \), the beam \( B \), the draft \( D \), and the block coefficient \( C_b \).

The numerical approach only involves linear potential flow computations for the ship at rest, i.e. for the wetted hull surface \( \Sigma^H_0 \), rather than for the mean wetted hull surface \( \Sigma^H \) of the ship at its actual position. This practical simplification stems from the fact that the sinkage and the trim are mostly determined by the pressure distribution over the lower part of the ship hull surface, and consequently are not highly sensitive to the precise position of the ship. A linear potential flow method is used in the numerical approach considered in [11].

Both the simple numerical approach and the even simpler experimental approach are found in [11] to yield realistic predictions of sinkage and trim at \( F \leq 0.45 \).

A practical approach for determining the drag of a typical freely floating monohull ship at a moderate Froude number \( F \leq 0.45 \) is considered here. Specifically, classical semiempirical relations for the friction drag, the viscous drag and the drag due to hull roughness are used, and the wave drag is evaluated via a practical linear potential flow method, as given in [12]. The drag is much more sensitive to the hull position than the sinkage and the trim (for a simple reason explained in section 2). Accordingly, the drag must be computed for a ‘dynamic’ ship hull surface \( \Sigma^H_{st} \) that accounts for sinkage and trim effects. Specially, the hull surface \( \Sigma^H_{st} \) can be chosen as the hull surface \( \Sigma^H \) that is predicted by the numerical approach, or as the hull surface \( \Sigma^H_{at} \) that is predicted by the even simpler experimental approach, as illustrated via numerical
computations in section 6.

Moreover, the drag of the hull surface $\Sigma_H^1$ and the (nearly identical) drag of the hull surface $\Sigma_H^a$ are significantly higher — and also much closer to experimental measurements for the Wigley, S60 and DTMB5415 models — than the drag of the hull surface $\Sigma_H^0$ of the ship at rest at high Froude numbers $F$ (within the constraint $F \leq 0.45$ considered here). These numerical results suggest that sinkage and trim effects, significant at Froude numbers $0.35 \leq F$, on the drag of a typical freely floating monohull ship can be well accounted for in a practical way that only requires linear potential flow computations, without iterative computations for a sequence of hull positions.

Hereafter, coordinates and flow variables are made nondimensional in terms of the gravitational acceleration $g$, the water density $\rho$, and the length $L$ and the speed $V$ of the ship. The Cartesian system of nondimensional coordinates $(x, y, z) \equiv (x, y, z)/L$ is attached to the moving ship. The $x$ axis is chosen along the path of the ship and points toward the ship bow. The undisturbed free surface is taken as the plane $z = 0$ and the $z$ axis points upward. The ship bow and stern are located at $x_b = (0.5, 0, 0)$ and at $x_s = (-0.5, 0, 0)$. The unit vector $\mathbf{n} \equiv (n_x, n_y, n_z)$ is normal to the hull surface $\Sigma_H$ and points outside the ship (into the water).

The present study summarizes the analysis and the main results of the practical approaches considered in [11] and [12] for estimating the sinkage and the trim, and their influence on the drag, for a common freely-floating monohull ship at moderate Froude numbers $F \leq 0.45$.

2 BASIC RELATIONS FOR THE SINKAGE AND THE TRIM

Hereafter, the vertical displacement of a ship hull surface $\Sigma_H$ from its position $\Sigma_H^0$ at rest, at midship, is called ‘midship sinkage’ and denoted as $H_m$. Similarly, the vertical displacement of $\Sigma_H$ at the ship bow and stern are denoted as $H_b$ and $H_s$, and called ‘bow sinkage’ and ‘stern sinkage’. Positive values of $H_m$, $H_b$ or $H_s$ correspond to downward vertical displacements at midship, at a ship bow or at a ship stern, respectively. The rotation of $\Sigma_H$ from $\Sigma_H^0$ is defined by the trim angle $\tau^\circ \equiv \tau^{\text{rad}}180/\pi$ where the angles $\tau^\circ$ and $\tau^{\text{rad}}$ are measured in degrees or in radians, or by the equivalent ‘trim sinkage’ $H^\tau$ defined as

$$2H^\tau \equiv L\tan(\tau^{\text{rad}}) \approx L\tau^{\text{rad}} \equiv L\tau^\circ \pi/180$$

(1)

Positive values of $\tau^\circ$, $\tau^{\text{rad}}$, $H^\tau$ correspond to a bow-up rotation.

The relations $H^s = H^m + H^\tau$ and $H^b = H^m - H^\tau$ hold. These geometrical identities determine the stern sinkage $H^s$ and the bow sinkage $H^b$ from the midship sinkage $H^m$ and the trim sinkage $H^\tau$ that are computed in the numerical approach considered in section 3. These relations readily yield

$$H^b = 2H^m - H^s \quad \text{and} \quad H^\tau = H^s - H^m$$

(2)

These relations determine the bow sinkage $H^b$ and the trim sinkage $H^\tau$ from the midship sinkage $H^m$ and the stern sinkage $H^s$ that are determined in section 4 by simple analytical relations obtained via an analysis of experimental measurements.

3 NUMERICAL DETERMINATION OF THE SINKAGE AND THE TRIM

The midship sinkage $H^m$ and the trim sinkage $H^\tau$, where positive $H^m$ and $H^\tau$ respectively correspond to a downward vertical displacement or a bow-up rotation of the ship hull surface
The terms \( C^z \) and \( C^{zx} \) in (3a) represent the nondimensional hydrodynamic lift and moment coefficients defined as

\[
(a_0, a_1, a_2) = \left( \frac{A_0}{L^2}, \frac{A_1}{L^3}, \frac{A_2}{L^4} \right) = \int_{\Sigma_0^H} (1, x, x^2) \, dx \, dy
\]

(3b)

The hydrodynamic pressure \( p \) is given by the Bernoulli relation

\[
p = \sqrt{(ny)^2 + (n^z)^2 \phi_t + (n^x)^2/2 - (\phi_t^2 + \phi_d^2)/2}
\]

(3d)

Here, \( \phi_t \equiv \partial \phi / \partial t \) and \( \phi_d \equiv \partial \phi / \partial d \) denote the velocity components along two unit vectors \( t \) and \( d \) tangent to the ship hull surface \( \Sigma^H \). These tangential velocity components are evaluated here via the Neumann-Michell (NM) theory expounded in [1] and [13].

The nondimensional hydrodynamic lift and pitch-moment coefficients \( C^z \) and \( C^{zx} \) given by (3c) where the mean wetted ship hull surface \( \Sigma^H \) is taken as the static wetted hull surface \( \Sigma_0^H \) are denoted as \( C_0^z \) and \( C_0^{zx} \). \( H^m \) and \( H^r \) given by (3a) where \( C^z \) and \( C^{zx} \) are taken as \( C_0^z \) and \( C_0^{zx} \) are similarly denoted as \( H_0^m \) and \( H_0^r \). The mean wetted hull surface that is obtained from the wetted hull surface \( \Sigma_0^H \) of a ship at rest via a translation \( H_0^m \) and a rotation \( H_0^r \) is denoted as \( \Sigma_0^H \), and the hydrodynamic coefficients \( C^z \) and \( C^{zx} \) given by (3c) with \( \Sigma^H \) taken as \( \Sigma_0^H \) are denoted as \( C_1^z \) and \( C_1^{zx} \). Similarly, \( H_1^m \) and \( H_1^r \) denote the sinkage \( H^m \) and the trim \( H^r \) determined from (3a) with \( C^z \) and \( C^{zx} \) taken as \( C_1^z \) and \( C_1^{zx} \).

Expressions (3c) for \( C^z \) and \( C^{zx} \) show that, except for a ship hull with large flare and rake angles, the upper part of a ship hull (where \( n^z \approx 0 \)) does not contribute appreciably to the sinkage and that the upper hull and the parallel midbody (where \( n^z \approx 0 \) and \( n^x \approx 0 \)) do not contribute much to the trim. Thus, the main contributions to the sinkage and the trim stem from the lower part of the ship hull surface. It can therefore be expected that the sinkage and the trim of a ship are relatively insensitive to the precise position of the ship hull, and can be realistically evaluated from the pressure distribution around the hull surface \( \Sigma_0^H \) of the ship at rest. This theoretical expectation is confirmed in [11] via numerical comparisons of the sinkage \( H^m \) and the trim \( H^r \) determined for the hull surfaces \( \Sigma^H \) taken as \( \Sigma_0^H \) or \( \Sigma_1^H \).

4 EXPLICIT APPROXIMATIONS FOR THE SINKAGE AND THE TRIM

The previously-noted fact that the sinkage and the trim of a ship predominantly stem from the pressure distribution over the lower part of the ship hull surface also suggests that the sinkage and the trim may not be highly sensitive to ‘details’ of the hull form (such as a transom stern) and the Reynolds number, and might reasonably be assumed to primarily depend on
the Froude number and basic hull form parameters (such as the beam/length ratio $B/L$ and the block coefficient $C_b$) that characterize the overall hull geometry. This theoretical conjecture is considered in [11] via an analysis of experimental measurements of sinkage and trim for 22 models of freely-floating monohull ships.

### 4.1 Experimental data base

Specifically, the experimental measurements of sinkage and trim for 22 models of monohull ships reported in publications are described and analyzed in [11].

The beam/length ratio $B/L$, the draft/length ratio $D/L$, the draft/beam ratio $D/B$ and the block coefficient $C_b$ for these 22 ship models vary within the relatively broad ranges

- $0.066 \leq B/L \leq 0.148$
- $0.029 \leq D/L \leq 0.071$
- $0.276 \leq D/B \leq 0.667$
- $0.397 \leq C_b \leq 0.6$

These ranges of variations of $B/L$, $D/L$, $D/B$ and $C_b$ correspond to a wide range of hull forms.

Fig.1 depicts the experimental measurements, for Froude numbers $F$ within the range $0.1 \leq F \leq 0.45$, of the midship sinkage $H^m$, the trim sinkage $H^\tau$, the stern sinkage $H^s$ and the bow sinkage $H^b$ for the 22 ship models. This figure shows that $H^m$ and $H^s$ mostly increase monotonically as the Froude number $F$ increases. However, the variations of the trim sinkage $H^\tau$ and the bow sinkage $H^b$ are more complicated. Accordingly, only the experimental measurements of the midship sinkage $H^m$ and the stern sinkage $H^s$ are analyzed. The bow sinkage $H^b$ and the trim sinkage $H^\tau$ can be subsequently determined from $H^m$ and $H^s$ via the relations (2).
4.2 Midship sinkage

The variations of the experimental measurements of the midship sinkage $H_m$ for the 22 models of monohull ships depicted in the top left corner of Fig.1 with respect to the Froude number $F$ and the four basic hull form parameters $B/L$, $D/L$, $D/B$ and $C_b$ are analyzed in [11]. This analysis shows the midship sinkage $H_m$ increases approximately like $F^2$ as $F \leq 0.45$ increases, is approximately proportional to $\sqrt{BD}$, and moreover increases as the block coefficient $C_b$ increases. Specifically, the detailed analysis of experimental measurements of the midship sinkage $H_m$ given in [11] shows that $H_m$ can be explicitly estimated in terms of the beam $B$, the draft $D$, the block coefficient $C_b$ and the Froude number $F$ via the simple analytical relations

$$H_m/\sqrt{BD} \approx F^2 C_m$$

where $C_m \equiv 0.9(C_b - 0.13)$ (4)

with an accuracy of 20% in most cases and 30% in nearly all cases of a wide range of monohulls.

4.3 Stern sinkage

The variations of the experimental measurements of the stern sinkage $H_s$ for the 22 models of monohull ships depicted in the bottom left corner of Fig.1 with respect to the Froude number $F$ and the four basic hull form parameters $B/L$, $D/L$, $D/B$ and $C_b$ are analyzed in [11]. This analysis shows that the stern sinkage $H_s$ increases approximately like the function $f$ defined as

$$f \equiv F_s^2 \sqrt{1 + F_s^8}$$

with $F_s \equiv F/0.33$ (5a)

as $F \leq 0.45$ increases, and is approximately proportional to $\sqrt{BD}$. The available experimental measurements of $H_s$ show no convincing correlations between $H_s$ and $C_b$. Specifically, the detailed analysis of experimental measurements of the stern sinkage $H_s$ given in [11] shows that $H_s$ can be explicitly estimated in terms of the beam $B$, the draft $D$, and the Froude number $F$ via the simple analytical relations

$$40 H_s/\sqrt{BD} \approx f$$

(5b)

with an accuracy 20% in many cases and 40% in most cases of a wide range of monohull ships, where $f$ is given by (5a).

4.4 Explicit analytical approximations

The midship sinkage $H_m$ and the stern sinkage $H_s$ can then be approximately determined by means of relations (4) and (5), and the bow sinkage $H_b$ and the trim sinkage $H^\tau$ can be determined from $H_m$ and $H_s$ via the relations (2). Thus, the analysis of experimental measurements considered in [11] determines $H_m$, $H_s$, $H_b$ and $H^\tau$ via the analytical relations

$$H_m \approx 0.9\sqrt{BD} (C_b - 0.13) F^2$$

(6a)

$$H_s \approx 0.025\sqrt{BD} F_s^2 \sqrt{1 + F_s^8}$$

with $F_s \equiv F/0.33$ (6b)

$$H_b = 2H_m - H_s$$

and $H^\tau \equiv L r^\tau \pi/360 = H_s - H_m$ (6c)

The trim angle $r^\tau$ in (6c) is measured in degrees. These relations explicitly determine the sinkage and the trim without flow computations, and are then particularly simple.
5 PRACTICAL EVALUATION OF THE DRAG

The nondimensional drag coefficient

\[ C^t \equiv \frac{D}{\rho V^2 L^2} \]  

(7)

is evaluated in [12] and here in a simple way, based on the classical Froude decomposition into viscous and wave components, as in Yang et al. (2013). Specifically, \( C^t \) is expressed as

\[ C^t = C^w + C^v + C^a \]  

(8)

where \( C^w \) represents the wave drag coefficient, \( C^v \) is the viscous drag coefficient for a smooth ship hull, and \( C^a \) accounts for the additional drag due to roughness.

The viscous drag \( C^v \) in (8) is expressed as

\[ C^v = (1 + k) C^f \]  

(9a)

where \( C^f \) and \( k \) are the usual friction drag coefficient and form factor. The friction drag \( C^f \) is evaluated via the ITTC 1957 formula

\[ C^f = \frac{A_H^2}{2L^2} \left( \frac{0.075}{(\log_{10} R_e - 2)^2} \right) \text{ where } R_e = \frac{V L}{\nu} \]  

(9b)

and \( A^H \) denotes the wetted area of the ship hull surface \( \Sigma^H \). The kinematic viscosity \( \nu \) is taken as \( 1.14 \times 10^{-6} \text{m}^2/\text{s} \) hereafter. The form factor \( k \) is estimated via the relation

\[ k = 0.6 \sqrt{\frac{\Delta}{L^3}} + 9 \frac{\Delta}{L^3} \text{ with } 0.05 \leq k \leq 0.40 \]  

(9c)

given in [14]. Here, \( \Delta \) denotes the displacement of the ship.

The roughness correction \( C^a \) in (8) is determined via the Bowden-Davison formula

\[ C^a = 10^{-4} \frac{A^H}{2L^2} R \text{ where } 4 \leq R \equiv 1050 \left( \frac{k_s}{L} \right)^{1/3} - 6.4 \leq 8 \]  

(10)

given in [15]. \( k_s \) characterizes the roughness of the hull surface, and \( k_s = 0.00015 \text{m} \) is used here.

The wave drag coefficient \( C^w \) is determined via integration of the pressure \( p \) at the hull surface \( \Sigma^H \), i.e.

\[ C^w = \int_{\Sigma^H} n^x p \, da \]  

(11)

where \( p \) is given by the Bernoulli relation (3d). The Neumann-Michell theory is used to compute the flow around the ship hull surface \( \Sigma^H \) and the related flow pressure \( p \). Expression (11) for \( C^w \) shows that the parallel midbody section and bottom part of a ship hull surface, where \( n^x \approx 0 \), contribute little to the wave drag, which mostly stems from the upper parts of the bow and stern regions where \( n^x \neq 0 \). The drag of a ship can therefore be expected to be much more sensitive to the precise position of the ship hull than the sinkage and the trim, which are mostly determined by the pressure distribution over the hull bottom as noted earlier.

\( A^H_0, A^H_1 \) and \( A^H_a \) denote the wetted areas of the hull surfaces \( \Sigma^H_0 \) of the ship at rest or the hull surface \( \Sigma^H_1 \) or \( \Sigma^H_a \) determined from the sinkage and the trim predicted by the numerical
Figure 2: Side views (left) and bottom views (right) of the wetted hull surfaces $\Sigma^H_0$ of the Wigley hull (top), the S60 model (middle) and the DTMB5415 model (bottom) approximated via 7,562 (Wigley), 11,542 (S60) and 12,586 (DTMB5415) flat triangular panels.

approach or the experimental approach considered in sections 3 and 4. Expression (9a) shows that differences among the wetted areas $A^H_0, A^H_1$ and $A^H_a$ yield differences among the friction drag coefficient $C^f$. The total drag coefficients $C^t_0, C^t_1, C^t_a$, the viscous drag coefficients $C^v_0, C^v_1, C^v_a$ and the wave drag coefficients $C^w_0, C^w_1, C^w_a$ correspond to the hull surfaces $\Sigma^H_0, \Sigma^H_1$ or $\Sigma^H_a$, respectively.

6 ILLUSTRATIVE APPLICATIONS FOR THREE SHIP MODELS

The simple methods for determining the sinkage, the trim and the drag of a freely-floating ship given in the foregoing are now applied to three ship models: the Wigley hull, the S60 model and the DTMB5415 model. The length $L$ of these ship models is 2.5m for the Wigley hull, 4m for the S60 model and 5.72m for the DTMB5415 model. Side and bottom views of the wetted hull surfaces $\Sigma^H_0$ for these three ship models are shown in Fig.2. Half of the hull surface $\Sigma^H_0$ is approximated via 7562, 11,542 or 12,586 flat triangular panels for the Wigley hull, the S60 model and the DTMB5415 model, respectively.

Fig.3 depicts the midship sinkage $H^m/D$ and the trim sinkage $H^\tau/D$ estimated via numerical approach (3), experimental approach (6) or experimental measurements for the three ship models at $0.1 \leq F \leq 0.45$.

Fig.3 shows that the numerical predictions for the hull surfaces $\Sigma^H_0$ or $\Sigma^H_1$ are nearly identical for the midship sinkage $H^m$, and do not differ significantly for the trim sinkage $H^\tau$. Moreover, these numerical predictions are in relatively good agreement with the experimental measurements. The numerical predictions for the hull surface $\Sigma^H_1$ are not closer to the experimental measurements of the trim sinkage $H^\tau$ than the numerical predictions for the hull surface $\Sigma^H_0$. This finding suggests that it is sufficient to compute the flow around the ‘static’ hull surface $\Sigma^H_0$, instead of the ‘dynamic’ hull surface $\Sigma^H_1$, for the purpose of predicting the sinkage and the trim of common monohull ships at Froude numbers $F \leq 0.45$. Fig.3 also shows that the simple analytical relations (6) yield predictions of the midship sinkage $H^m$ and the trim sinkage $H^\tau$ that are in relatively good agreement with the numerical predictions, as well as the experimental measurements.

Fig.4 depicts the theoretical predictions of the total drag $C^t$, the wave drag $C^w$ and the viscous drag $C^v$ that correspond to the hull surfaces $\Sigma^H_0, \Sigma^H_1$ and $\Sigma^H_a$ for the Wigley hull, the S60 model and the DTMB5415 model. The experimental measurements $C^t_e$ of the total drag $C^t$ that are also shown in Fig.4 correspond to freely-floating ship models, and are determined via

$$C^t_e = C^r + C^v_0$$

(12)
Figure 3: The midship sinkage $H^m/D$ (left) and the trim sinkage $H^T/D$ (right) for the Wigley hull (top), the Series 60 model (center) and the DTMB5415 model (bottom). Experimental measurements (Exp.) are shown together with the predictions given by the analytical relations (6) obtained from an analysis of experimental measurements (Appr.) and the numerical predictions given by the Neumann-Michell theory applied to the hull surfaces $\Sigma^H_a$ or $\Sigma^H_1$.

Figure 4: The left side depicts the wave drag $C^w$ and the right side depicts the viscous drag $C^v$ for the Wigley hull (top), the S60 model (center) and the DTMB5415 model (bottom). Experimental measurements (Exp.) are shown together with the theoretical predictions (8)-(11) applied to the hull surfaces $\Sigma^H_a$ or $\Sigma^H_1$.

where $C^w$ denotes the residual drag of the freely-floating ship model and $C^v_w$ is the viscous drag of the hull surface $\Sigma^H_0$ of the ship at rest.

Fig.4 shows that differences among the theoretical drag coefficients $C^t$, $C^w$ and $C^v$ for the hull surfaces $\Sigma^H_1$ and $\Sigma^H_a$ are practically negligible. Moreover, the total drags $C^t$ of the hull surfaces $\Sigma^H_1$ and $\Sigma^H_a$ are in reasonable overall agreement with the experimental measurements $C^t_a$. Fig.4 also shows that differences between the theoretical drag coefficients $C^t$, $C^w$, $C^v$ for the hull surfaces $\Sigma^H_1$ or $\Sigma^H_a$ and the hull surface $\Sigma^H_0$ of the ship at rest are fairly small for $F < 0.25$, but increase rapidly for $0.25 < F$. Sinkage and trim effects on the drag can then be ignored for $F < 0.25$, but can be significant for $0.25 < F$.

Experimental measurements, denoted as $C^t_e$, of the total drag $C^t$ for a freely-floating ship are now compared to the corresponding theoretical predictions $C^t_0$ or $C^t_1$ for the ship hull surfaces $\Sigma^H_0$ or $\Sigma^H_1$. The relative differences between the experimental measurements $C^t_e$ and the corresponding theoretical predictions $C^t_0$ or $C^t_1$ for the ship hull surfaces $\Sigma^H_0$ or $\Sigma^H_1$ are

$$e^t_0 = (C^t_e - C^t_0)/C^t_0 \quad \text{and} \quad e^t_1 = (C^t_e - C^t_1)/C^t_1$$  \hspace{1cm} (13)

The relative errors $e^t_0$ and $e^t_1$ are associated with the hull surface $\Sigma^H_0$, which ignores sinkage and
Fig. 5: Relative errors $e^t_0$ and $e^t_1$, and corresponding smoothing spline fits, between experimental measurements of the total drag $C_t$ and theoretical predictions for the hull surface $\Sigma^H_0$ of a ship at rest or the hull surface $\Sigma^H_1$ that accounts for sinkage and trim effects.

Thus, the errors $e^t_0$ and $e^t_1$ provide a basis for validating the theoretical method considered here to account for the influence of sinkage and trim on the drag of a freely-floating ship.

Fig. 5 depicts the relative errors $e^t_0$ and $e^t_1$ for the Wigley, S60 and DTMB5415 models within $0.25 < F < 0.45$ for which sinkage and trim have a significant influence on the drag. The dashed and solid lines in Fig. 5 are smoothing spline fits that correspond to the experimental values of $e^t_0$ (squares) or $e^t_1$ (circles). The errors $e^t_0$ increase for $0.28 < F$ and are significant for $0.35 < F$. Indeed, the errors $e^t_0$ are larger than 10% for $0.4 < F$. The errors $e^t_1$ are much smaller than the errors $e^t_0$ for the Wigley hull and the DTMB5415 model at $0.32 < F$, and for the S60 model at $0.35 < F$. Specifically, the errors $e^t_1$ vary within $\pm 2\%$ for the Wigley and S60 models in the Froude number ranges $0.32 < F$ or $0.35 < F$, and vary within $\pm 7\%$ for the DTMB5415 model at $0.32 < F$.

7 CONCLUSIONS

The influence of sinkage and trim on the drag of a common freely floating monohull ship has been considered. The comparisons of experimental measurements and theoretical computations reported for the Wigley, S60 and DTMB5415 models suggest that the sinkage and the trim experienced by common monohull ships are small, and have limited influence on the drag, for Froude numbers $F < 0.25$. However, the sinkage and the trim, and their influence on the drag, increase rapidly for $0.25 < F$, and are significant for the highest value $F = 0.45$ of the Froude number range $0.1 \leq F \leq 0.45$ considered here. Sinkage and trim effects should then be considered within the design process, arguably even at early design stages and for hull form optimization.

Accordingly, practical methods suited for routine applications to ship design have been considered in [11] and here to determining the sinkage, the trim and the drag. Specifically, the sinkage and the trim are determined via two simple alternative methods, previously considered in [11]. These two methods yield predictions of sinkage and trim that do not differ greatly, and moreover are in reasonable agreement with experimental measurements for a broad class of monohull ships. The drag is similarly evaluated here in a simple way, based on the classical Froude decomposition into viscous and wave components.
One of the two simple alternative methods considered in [11] and here to determine the sinkage and the trim is a numerical method. This method only involves linear potential flow computations (the Neumann-Michell theory is used) for the wetted hull surface $\Sigma^H_0$ of the ship at rest, instead of the ‘dynamic’ hull surface $\Sigma^H_1$, which is determined from flow computations for the hull surface $\Sigma^H_0$. This notable simplification stems from the fact that the sinkage and the trim are primarily determined by the pressure distribution over the lower part of the ship hull surface, and therefore are not highly sensitive to the precise position of the ship.

The other method considered in [11] and here to determine the sinkage and the trim is based on an analysis of experimental measurements (for 22 ship models). This alternative method yields explicit analytical relations for the sinkage and the trim in terms of the Froude number $F$, the beam $B$, the draft $D$, and the block coefficient $C_b$, and thus requires no flow computations.

As already noted, the drag is also estimated here in a simple way, based on the classical Froude decomposition of the drag into viscous and wave components. The wave drag is largely determined by the pressure distribution on the bow and the stern of a ship, and is therefore much more sensitive to the precise position of the ship hull than the sinkage and the trim, which are mostly determined by the pressure distribution over the hull bottom as already noted. This basic difference explains why the sinkage and the trim of a ship can be realistically estimated from flow computations around the hull surface $\Sigma^H_0$ of the ship at rest, whereas the drag must be evaluated for a ‘dynamic’ ship hull surface $\Sigma^H_{st}$ that accounts for the sinkage and the trim experienced by the ship.

However, the hull surface $\Sigma^H_{st}$ does not need to be very precise. Indeed, a main result of the numerical computations reported here for the Wigley, S60 and DTMB models is that the hull surface $\Sigma^H_a$ defined by the explicit analytical relations (6) and the hull surface $\Sigma^H_0$ determined from potential flow computations for the hull surface $\Sigma^H_0$ of the ship at rest have nearly identical total drag coefficients $C_t$. Moreover, $C_t$ determined from the hull surface $\Sigma^H_a$ or $\Sigma^H_0$ are significantly higher than the drag $C_t$ predicted for the hull surface $\Sigma^H_0$ of the ship at rest.

Moreover, and most importantly for practical applications, the drag coefficients predicted for the hull surfaces $\Sigma^H_a$ or $\Sigma^H_0$ are much closer to experimental measurements than the drag of the hull surface $\Sigma^H_0$ of the ship at rest for the Wigley, S60 and DTMB5415 models at Froude numbers for which sinkage and trim effects are significant, as illustrated in Fig.5.

This finding provides a partial validation of the simple approach considered here, and suggests that the influence of sinkage and trim on the drag of a freely floating monohull ship at $F \leq 0.45$ can be determined in a very simple way well suited for routine applications to design, including at early stages and for optimization. In particular, if the analytical relations (6) are used to estimate the sinkage and the trim, prediction of the drag of a freely floating ship only requires a computation of the flow around the hull $\Sigma^H_a$, i.e., a single (linear potential) flow computation per Froude number.

REFERENCES


STEPS TOWARDS A SELF CALIBRATING, LOW REFLECTION NUMERICAL WAVE MAKER USING NARX NEURAL NETWORKS

Pál Schmitt *

*Marine Research Group,
Queen’s University Belfast,
Marine Laboratory, BT22 1PF Portaferry,
Northern Ireland
p.schmitt@qub.ac.uk

Key words: Numerical Wave Maker, Numerical Wave Tank, Computational Fluid Dynamics (CFD), Neural Network, non-linear autoregressive exogenous input (NARX)

Abstract. Numerical wave-makers are important for accurate simulations of marine two-phase flows. One important aspect of the suitability is the ability to recreate a given surface elevation time trace at a given location in the tank. This paper presents first applications of NARX neural network techniques for the calibration of wave tanks. Preliminary results indicate good results can be achieved even for highly non-linear waves.

1 INTRODUCTION

The accurate application of computational fluid dynamics methods to problems in marine and offshore engineering depend largely on the ability to create accurate representations of gravity water waves in the computational domain. Many different methods have been proposed to create and absorb waves in simulations, but as in experimental test facilities, it can still be difficult to create an exact target time trace of surface elevation at a certain position [6]. Most experimental facilities rely on tank transfer functions, which describe the relation between the wave amplitude and the wave maker input for each frequency component used to generate a time series [5]. By decomposing a surface elevation trace, the phase lag and amplitude of each individual component can be obtained and a wave trace can be created. This method is of course limited to a flat bottom and cannot take into account non-linear interaction between waves. In addition, numerical tools are also often used to simulate only very short time traces, methods based on spectral separation of individual components might not be readily applicable.

Neural networks, a type of machine learning tool based to some extend on biological examples, have been shown to work well in many cases where complex non-linear
interactions are of interest and sufficient training data is available [1]. Applications in coastal engineering have ranged from the prediction of ocean wave parameters to tidal levels, damages of coastal structures, the prediction of seabed liquefaction, changes of the near-shore morphology and the prediction of wave heights and periods [3, 4].

Only one application of neural networks to the calibration of an experimental wave maker is known to the author [2]. Neural networks were used to predict the amplitude for regular waves and the wave energy within specific wave period bins for irregular waves. However, this work did not consider calibration of a given surface elevation trace.

2 APPLICATION TO A WAVEMAKER

The numerical wave tank used in this paper is based on the momentum source wave maker presented in [6]. The domain is 13.4m long and 1m high. The water level is 0.338m. Dissipation zones stretch about three meters at both ends of the tank. The wave maker zone is located 4.1-4.6m from the left and reaches from the bottom to the water surface. The wave probe at the calibration position is located at x=8.487m.

A nonlinear autoregressive exogenous model (NARX) neural network is used to find the wavemaker input that creates a target surface elevation. NARX neural networks belong to the recurrent neural network types, which means that the predicted value of the output is also used as an input in the network. Those types of networks have been developed specifically for the simulation of time dependent, multiple input prediction problems. Besides the number of hidden layers the lag between the recurrent input and the number of exogenous inputs to be used must be chosen. All simulations presented in this paper used the following algorithm:

• The feedback delay was chosen based on the lag of significant autocorrelation of the output signal

• The input delay was chosen from the lag of the significant cross correlation of the input and output signal

• The number of hidden layers was then determined by increasing the number of layers, if no more improvement in training results was seen, the last number of layers was used

The calibration process for a desired surface elevation begins with running a random input to the wave maker for 30 seconds. The obtained surface elevation is then used with the known wave maker input to predict a suitable wave maker input for the next simulation. The simulations are always started at each run from still water condition. It would be expected that with a larger number of training data each successive run produces better results. As an error measure the mean square error (MSE) and the mean square relative error (MSRE) are used in this paper.
3 DELAYS

[1] has described some issues with long-term memory of Neural Networks and some improvement NARX methods offer. Tests revealed that the networks had significant problems modelling the relationship between wave maker input and surface elevation. It was found that shifting the surface elevation by approximately the time the wave needed to travel across the tank significantly improved the predictions.

Figure 1 presents the development of MSE and MSRE over successive simulations. It would be expected, that each run reduces the error since more training data becomes available. A simulation without any delay, that is feeding the wave maker input and surface elevation directly into the simulation, yields very bad results. MSE and MSRE stay almost constant or even increase with more available data. Increasing the delay to 2 seconds results in even 2 or 1.3 times larger MSE or MSRE respectively. The errors can be seen to decrease significantly for a delay of 3 seconds. MSE and MSRE now also show a decrease over the number of runs. Figure 2 shows the target surface elevation time trace and the result achieved after 40 runs. For the most part, between 2 and 8 seconds, the agreement is remarkable. It should also be noted, that the waves used in this example are quite extreme, with a wave height of about half the water-depth.

4 RESTART AS INPUT

Although these first results are encouraging, restarting each iteration from still water might cause issues. A restart is required to accommodate the delay feature described above, but it seems plausible that better results could be obtained if the information about restarts was used in the learning process. The network was thus extended to two inputs. One is as before the surface elevation, but now a second auxiliary variable is used to indicate the beginning of a new run. A comparison of the errors and actual results of surface elevation are shown in figures 3 and 4. It can be observed that the two input
version, that is the neural network with knowledge of the restart, show no improvement during the first 20 runs. This might be attributed to the lack of actual training data for the restart variable during the first runs. Between 30 and 40 runs the results are somewhat better than the previous version. Interestingly, for more than 40 runs, the absolute and the relative mean square error increase again and remain higher for the two input version. This result requires further investigation.

Figure 2: Target and achieved surface elevation trace after 49 runs.
Figure 3: Comparison of MSE and MSRE over runs for single and two-input runs.

Figure 4: Target and achieved surface elevation trace after 40 runs for the 1 and 2 Input set-up.
5 CONCLUSIONS

With a relative mean square error of less than 0.5% after only a few iterations NARX neural networks have demonstrated to be a valuable tool for the calibration of time traces in wave tanks. It is believed that further investigations, especially in the optimal choice of feedback and input delays and number of hidden layers will further improve results.

6 ACKNOWLEDGEMENTS

Pál Schmitt’s was funded from the European Unions Horizon 2020 research and innovation programme under grant agreement No 654438.

REFERENCES


SYSTEMATIC ANALYSIS OF MESH AND MESHLESS CFD METHODS FOR WATER IMPACT PROBLEMS

LUCA BONFIGLIO*, STEFANO GAGGERO†, ALDO PAPETTI†, GIULIANO VERNENGO† AND DIEGO VILLA†

*MIT Sea Grant College Program
Massachusetts Institute of Technology
Cambridge, 02139-4307, MA, USA
e-mail: bonfi@mit.edu

† Dept. of Electric, Electronic and
Telecommunication Engineering and Naval Architecture (DITEN)
University of Genova
Genova, 16145, Italy
e-mail: stefano.gaggero@unige.it, giuliano.vernengo@unige.it, diego.villa@unige.it

Key words: Slamming, Violent Impact Hydrodynamics, Impact Forces, Free Surface Deformation, Unsteady Reynolds Averaged Navier Stokes (URANS), Smoothed Particle Hydrodynamics (SPH)

Abstract. Two types of numerical simulations for the hydrodynamic solution of water entry problems are performed and systematically compared in order to highlight their peculiarities: a viscous Unsteady Reynolds Averaged Navier Stokes (URANS) based on Volume of Fluid (VoF) approach and a meshless Smooth Particle Hydrodynamics (SPH) solver. In both cases open-source software have been chosen. The numerical solutions from the two proposed CFD methods are verified against experimental measurements. The systematic analysis is performed considering a prescribed motion of the wedge aiming to a better understanding of the effect of the model set-up on the prediction of both local and global field variables. Even if a both codes shows a high ability to capture the global physics of the problem, due to the strongly non-linear dynamic of the body-flow interactions involved in impact problems, the local pressure peaks can be hardly predicted if the numerical method is not suitably tuned for the specific problem. Finally the two approaches are performed also for a free falling simulation, comparing the numerical predictions to the available experimental drop tests results.

1 INTRODUCTION

Due to the strongly non-linear nature of the involved phenomena, impact hydrodynamics of falling bodies is a good test case to assess the capabilities of Computational Fluid Dynamics (CFD) solvers. Also, this is a very interesting problem in naval architecture since it is directly related to seakeeping, slamming loads and even more complex coupled analysis such as those
for ship springing and whipping generally resulting in various types of structural damages. The simpler experiment for such a class of problem consists of a drop test on symmetric or asymmetric bodies, such as wedges, cylinders or ship-like sections, usually meeting the bi-dimensional simplification. The first theoretic formulations belong to Von Karman [1] and Wagner [2] valid for small local wedge angles. Later Dobrovol’skaya [3] provides the so-called similar solution for wedges forced to impact the free surface at constant speed that needs to be solved numerically since given in implicit form. More recently, a great impulse to the numeric solution of the problem has been given by Faltinsen who develops solutions based on different approaches such as non-linear Boundary Element Method with a jet model (see for instance [4] and [5]) or matched asymptotic expansions technique [6]. Considering other numerical techniques, mesh-based Unsteady Reynolds Averaged Navier-Stokes (URANS) and mesh-less Smoothed Particle Hydrodynamics (SPH) methods are gaining higher levels of maturity both as formulations and from the hardware-related side to be considered for both research and practical applications [7]. However there are still some open questions for example on the effects of the model set-up on the accuracy and stability of the searched solution and on the possibility to generalize the obtained results. Many examples of solutions of the wedge impact problem using both methods can be found in literature (see among the other [9], [8], [10], [11], [12], [13] and [14]). Some preliminary analysis on the influence of some model parameters on the solution of wedges impact have also been recently given in [15] and [16]. The proposed numeric study aims exactly at a better understanding of the effects of the simulation set-up on the hydrodynamics solution of impact-related problems. This is achieved by a systematic analysis of some the most relevant model parameters of both CFD methods; both the local pressures and the global forces on the body are extracted from the simulations and compared to available experimental measurements.

The two methods, namely the URANS and the SPH are briefly described and results are shown separately. A final comparison among the two methods is performed and discussed in the light of the different formulations.

2 SELECTED TEST CASE FOR VALIDATION

All the computations presented in the study refer to the symmetric wedge tested by [17] with wedge angle $\alpha = 25^\circ$. The test reproduced in the systematic numeric analysis of the two CFD methods corresponds to a free falling from an initial height of $h_{\text{Drop}} = 1.3m$ with the mass of the wedge equal to $m_W = 94kg$. In order to speed-up the analysis, all the computations have been performed using the vertical displacements of the wedge measured during the drop test and provided in the referenced paper; hence, the simulations have been ran for a reduced range of time with respect to the physical duration of the experiments, focusing on the impact of the wedge into the water and excluding part of the initial falling and of the tail of the drop test.

The tested wedge have been equipped with twelve pressure transducer uniformly spaced along the face of the wedge itself as schematically shown in Figure 1. Due to the experimental set-up this test can be considered as bi-dimensional.

3 DESCRIPTION OF THE NUMERICAL METHODS

Two codes have been explored to tackle the wedge impact problem. The first is a mesh-based code which solves the viscous flow motion by means of the classical Finite Volume approximation
(FV). The second one is a mesh-free code based on the discretization of the fluid flow by means of smooth discrete particle (SPH). Both codes have the ability to solve the 6-DoF dynamic (for the present case only 1-DoF is considered) or to constrain the wedge to move following a prescribed path. The second approach has been preferred to analyze the solver parameters in terms of force and pressure results neglecting their influence on the wedge motion. The selected path has been extracted by the original paper [17]. After the selection of the proper model set-up the free falling simulation has been compared with the experimental results.

### 3.1 MESH-BASED APPROACH: URANS

An open-source mesh-based viscous code has been used in the present paper to be compared with the mesh-free approach. The viscous code is a part of the OpenFOAM framework which is a general purpose CFD solver based on the finite volume approximation. A modified version of the classical Navier-Stokes flow system of equations in its unsteady form is solved to tackle the wedge entrance problem. Indeed, this problem is complicated by the presence of a highly distorted free surface.

Numerically, the presence of the two fluids can be taken into account in different ways. Here it is exploited by means of the Volume of Fluid (VoF) approach. This method is based on the flow-mixture theory which add a new scalar field ($\alpha$) that represents the amount of liquid (or gas) in any point inside the domain. Consequently all the fluid properties will be further defined by a weighted average of the two considered fluids properties (air and water). The subscript l,g,m respectively stand for liquid, gas and mixture. They are summarized in Eq. (1).

\[
\rho_m(x,t) = \rho_l \cdot \alpha + \rho_g \cdot (1 - \alpha) \\
\mu_m(x,t) = \mu_l \cdot \alpha + \mu_g \cdot (1 - \alpha) 
\]  

The $\alpha$ field is further transported in time through a purely convective equation, which takes into account the mass conservation of each phase of the mixture considering their immiscible nature. The new system of equations, reported in Eq. (2), presents a similar form of the starting N-S ones referred to the whole mixture.
Figure 2: Multi-block structured mesh arrangement (coarser one)

\[
\begin{align*}
\nabla \cdot U_m &= 0 \\
\frac{\partial U_m}{\partial t} + \nabla (U_m \otimes U_m) &= -\frac{\nabla p_m}{\rho_m} + \nu_m \nabla^2 U_m \\
\frac{\partial \alpha}{\partial t} + \nabla (U\alpha) &= 0
\end{align*}
\]

Numerically the equations are discretized using a first order time scheme and a second order accurate schemes for spatial derivatives. In particular the convective equation of the Volume of Fluid is solved by means of the ad-hoc developed Predictor-Corrector Semi-Implicit Multidimensional Universal Limiter for Explicit Solution (MULES) scheme: it was properly designed to solve the phase equation ensuring the boundedness of the void fraction, increasing also the time stability (i.e. allowing for Courant numbers higher than 1) compared to the purely explicit version. The proposed schemes have been selected as a good compromise between accuracy and numerical stability of the solver. Regarding the choice of the order for time derivatives approximation, a preliminary analysis, not fully reported here for shortness, has been performed. It demonstrated that the simulation adopting the second order scheme provided quite similar results, both in terms of global force and pressure signals on the wedge, with respect to the first order approximation. Simulations with second order schemes, however, suffered of a lower stability and numerical noise which definitely headed towards the first order approximation.

Thanks to nature of the performed tests, the problem was simplified to a 2-D representation of the wedge with purely bi-dimensional flow features. This simplification (widely adopted for this type of problems [4]) reduced of several order the computational effort required to solve the flow field without any significant loss of reliability compared to almost 2-D measurements.

The mesh has been arranged by using of the OpenFOAM utility blockMesh, which generates structured multi-block meshes. An example (with the coarser mesh) is reported in figure 2. Due to the body-fitted structure of the mesh (i.e. no overlapping) the motion of the wedge was simply achieved by a rigid translation of the entire computational domain. This simple approach presumes an infinite depth of the tank, which influence on results, as shown in the following discussion, seems negligible. Based on these considerations, table 1 summarizes the
appropriate boundary conditions. A set of different analyses have been carried to identify the

Table 1: Boundary conditions for the mesh-based code

<table>
<thead>
<tr>
<th></th>
<th>Pressure</th>
<th>Velocity</th>
<th>α</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wedge</td>
<td>fixedFluxPressure</td>
<td>noslip</td>
<td>( \frac{\partial \alpha}{\partial n} = 0 )</td>
</tr>
<tr>
<td>Top</td>
<td>( p = 0 )</td>
<td>( \frac{\partial U}{\partial n} = 0 )</td>
<td>( \frac{\partial \alpha}{\partial n} = 0 )</td>
</tr>
<tr>
<td>Symmetry</td>
<td>symmetry</td>
<td>symmetry</td>
<td>symmetry</td>
</tr>
<tr>
<td>Side</td>
<td>( p = 0 )</td>
<td>( \frac{\partial U}{\partial n} = 0 )</td>
<td>( \frac{\partial \alpha}{\partial n} = 0 )</td>
</tr>
<tr>
<td>Bottom</td>
<td>( \frac{\partial p}{\partial n} = 0 )</td>
<td>( \frac{\partial U}{\partial n} = 0 )</td>
<td>( \frac{\partial \alpha}{\partial n} = 0 )</td>
</tr>
</tbody>
</table>

Figure 3: Details of the various meshes used for RANS sensitivity analysis.

more suitable numerical parameters of the model. In particular two aspects have been investigated: the influence of the simulation time step (i.e. maximum allowable Courant number) and the mesh size. The first analyzed aspect is the influence of the simulations time step. Due to the shock nature of the problem, an adaptive time-step approach has been preferred. This method saves time during the smooth part of the motion (as the initial free falling in air of the wedge) where longer time intervals can be adopted, without affecting any aspects of the solution; switching to smaller time steps when the impact occurs to ensure time accurate calculations of the unsteadiness and of the interaction with water. The selection of the time-step is driven by the maximum allowable Courant number, which has a strong correlation with local flow velocities and to the smaller cell size inside the domain. Starting from a reference mesh arrangement (Level 2 of figure 3), five different Courant numbers have been considered, ranging from 5 to 0.1. As expected, when a Courant number higher than the unity is considered, the solution becomes unstable. On the contrary, as reported in figure 4, the other time selections give qualitatively the same results in terms of maximum pressure peaks (figure 4a) and global forces (figure 4b), even if the computational effort increases of one order of magnitude. These results show that the time-step selection is driven only by the stability constraint, therefore for further simulations a Courant number equal to 0.5 was selected.

The second addressed aspect concerns the mesh size. Three refinements approaches have been
considered. At first, a global refinement has been performed halving each time the mesh edges, therefore increasing the number of cells by four times (2-D case). In figure 5 the results for the different meshes, starting from the coarser (with only 9635 cells) up to the finer (with 2.47M cells), are reported. The recorded force signals (figure 5b) are almost equal, except in correspondence to the first impact. In this interval the coarser meshes, due to the free surface smearing, undergoes a slower and anticipated increment of force. These results, however, demonstrate the negligible influence of the mesh size on the global force experienced by the wedge. A completely different conclusion is observed for what regards the pressure peaks on the wedge surface. Figure 5a shows a comparison of the maximum recorded pressure for each sensor. It is clear that even if a very fine mesh has been adopted, the mesh independence has not been reached.

Based on these initial results, different refinement approaches have been proposed. In particular, cells have been clustered in correspondence of the wall boundary layer, using higher mesh gradings in the wall normal direction. This technique does not significantly affect the total cell count by adding cells only locally. Therefore the refinements (namely -a, -b, -c) have been considered starting form the mesh named Level 3. Figure 6a summarizes the results. Again a strong dependence of the pressure intensity by the near wall mesh size has been found, especially in correspondence of the first five pressure probes. Even if this method seems to be able to perform
more accurate calculations with a small increment of computational costs, the selected refine-
ment approach (which refine the cell only in one direction) produces cells with highly aspect ratio
which may impair the robustness of the analysis. In particular, this could pose no-negligible
issues on the solution stability, especially if further refinements are requested. The final set of
refinements has been adopted to overcome this problem. It consists on a recursive isotropic
halving of the cells localized only in correspondence of the wall boundary, therefore generating a
high quality mesh, in terms of aspect ratio, with a lower impact on the total cell count. Starting
form the Level 2 mesh, four refined configurations (namely -R1, -R2, -R3 and -R4) have been
arranged. The finer mesh (Level 2-R4) consists of about 1M cells and grants, at the wall, a
first cell prism height that is a quarter of the cell size achieved with the global refinements
(Level 5 with about 2.47M cells). With this refinement strategy, a certain grid independence is
achieved: pressures from meshes Level 2-R3 and Level 2-R4 are very similarly predicted, also in
correspondence to the first probes that showed, in the previous analyses, the highest sensitivity
to mesh variations. In conclusion, if the focus is on local pressure distribution on the wedge
surface, an enough finer mesh near the wall should be used, taking care of the mesh quality.
For the present test case a squared near wall cell with edges of 0.8mm (Level 2-R3, 275k cells)
can be considered as a good compromise. On the contrary, if the focus is only on the prediction
of global force exerted by the wedge, a regular mesh without the local, isotropic refinements at
wall (Level 2, 38k cells) with the edge length of 13mm, ensures sufficient accuracy.

As previously reported all the sensitivity analyses have been carried out imposing the wedge
motion extracted by the original paper. Afterwards, the final mesh setup (Level 2-R3) has been
used to perform a free-falling simulation where the vertical motion has been solved at each time-
step. In figure 7a the pressure peaks from the numerical simulations (imposing or solving the
motion) are compared with the experimental measurements. In parallel, also the experimental
and the computed motion are shown in figure 7b. The two numerical simulations differ both
in terms of pressure peaks and motion. The motion discrepancies, which are directly connected
also with the pressure differences, can be due to the difficulties in extracting the measured law
of motion that is given, in [17], only graphically. Anyway, non negligible differences with the ex-
perimental data are highlighted in terms of pressure behavior, even if the global peak is quite in
good agreement with the measured one (lower than 5%). This last discrepancy can be partially
Figure 7: Comparison of the imposed falling law and the free falling simulations with experimental measurements

justified by the 2-D simplification adopted for all the numerical simulations. Instead, neglecting the border effect in span-wise direction, a higher total force should be computed in the computation, which consequently make the wedge penetration velocity lower. On the contrary this border effect can be considered negligible related to the pressures experimentally recorded in the center line of the wedge. As previously reported by [3], [18], [19], the velocity penetration is directly connected with the pressure peak behavior and intensity.

3.2 MESH-LESS APPROACH: SPH

An open-source SPH solver [20] originally formulated by [21] has been selected for the proposed analysis. It is specifically designed to exploit GP-GPU computation allowing for relatively fast computations using a great number of particles (depending of course on the GPU hardware installed).

According to the SPH method, the equations of the fluid dynamics are solved by a mesh-free Lagrangian approach based on particles description. Each particle carries all the relevant field information and is used for the numerical integration of the PDEs for the mass and momentum conservation. The kernel approximation involves the representation of a field variable (e.g. \( V \), \( \rho \) or \( p \)) and its derivatives in a continuous integral form by a suitable kernel function; the particle approximation step refers to the discretization process of the computational domain that is redefined by an initial distribution of discrete particles; according to this discrete model, field variables on a particle are computed by approximation using the nearest neighbor particles. Giving a function \( f(x) \) its integral form is expressed according to Eq. (3):

\[
f(x) = \int_{\Omega} f(x') \delta(x - x') dx'
\]  (3)

Using a kernel function \( W(x - x', h) \) that depends on the so-called smoothing length \( h \), Eq. (3) can be written in terms of Kernel approximation as in Eq. (4); this kernel, that ideally would be a Dirac function, is substituted by some analytic formulations that in any case vanish for separations greater than \( kh \). A cubic spline kernel function (Eq. (5)) is selected for the present
\[ \langle f(x) \rangle = \int_{\Omega} f(x') W(x - x', h) dx' \]  
\[ W(r, h) = \frac{7}{4\pi h^2} \begin{cases} 
\frac{3}{2} q^2 + \frac{3}{4} q^3 & 0 \leq q \leq 1 \\
\frac{1}{4} (2 - q)^3 & 1 \leq q \leq 2 \\
0 & q \geq 2 
\end{cases} \]  

Even if water is an incompressible fluid, assuming it is weakly compressible allows for pressure computation from the state equation (Eq. (6)) instead of solving for the Poisson’s equation.

\[ p = \frac{c_0^2 \rho_0}{\gamma} \left( \frac{\rho}{\rho_0} \right)^\gamma - 1 \]  

In Eq. (6), \( c_0 \) is the sound speed ranging from 50 m/s up to 250 m/s to ensure \( \text{Mach} < 0.1 \) and \( \gamma \) is a constant assumed equal to 7.

According to the SPH formulation, both the mass and the momentum conservation laws, Eq. (7) and Eq. (8) respectively, are written in terms of particles approximation as follows:

\[ \frac{d\rho_i}{dt} = \sum_{ij} m_j (u_i - u_j) \nabla_i W_{ij} \]  
\[ \frac{du_i}{dt} = -\sum_{ij} m_j \left( \frac{P_i}{\rho_i^2} - \frac{P_j}{\rho_j^2} + \Pi_{ij} \right) \nabla_i W_{ij} + g \]  

The summation over the two indexes \( i \) and \( j \) accounts for particles interactions; \( m, u \) and \( P \) are the particle mass, velocity and the pressure at a particle respectively. \( \Pi_{ij} \) is a force contribution used to avoid tensile instabilities, defined according to the following (9):

\[ \Pi_{ij} = \begin{cases} 
\alpha \mu_{ij} \bar{c}_{ij} r_{ij} & u_{ij} r_{ij} < 0 \\
0 & u_{ij} r_{ij} > 0 
\end{cases} \]  

being the artificial viscosity coefficient \( \alpha \) the main parameter that controls this additional force term.

First a preliminary analysis has been carried out in order to select the most suitable dimensions of the computational domain that needs to be modified with respect to the physical domain used for the experiments to cut off possible reflections of the pressure waves propagating in the fluid. A 2D tank 6m wide and filled for 3m depth has been used. The four significant parameters of the SPH model listed in Table 2 have been considered for the proposed systematic analysis. The possible values assigned to each of them have been selected based on the suggested values for such a class of problems. The peak pressures measured at the twelve probe locations and the time-histories of the total vertical forces on the wedge are shown in Figures 8, 9, 10, 11 and 12 for CFL, \( C_S \), \( d_P \), \( \alpha \) and \( C_{\text{Sound}} \) variations respectively. The global forces prediction is generally more stable with respect to parameter variations, being the oscillations of the peaks always close to 13kN, occurring in a time lapse of 2/100s. The peak pressure computation rises more issues; its
Table 2: SPH model parameters and ranges used in the systematic analysis.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFL</td>
<td>[0.1; 0.2; 0.5]</td>
</tr>
<tr>
<td>$C_S$</td>
<td>[0.8; 1.0; 1.2]</td>
</tr>
<tr>
<td>$d_P$</td>
<td>[$1.0e^{-03}$; $1.5e^{-03}$; $2.0e^{-03}$]</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>[0.01; 0.05; 0.10]</td>
</tr>
<tr>
<td>$C_{\text{Sound}}$</td>
<td>[20; 25; 30]</td>
</tr>
</tbody>
</table>

Figure 8: Peak pressures (left) and global vertical forces on the wedge (right) with respect to variations of the Courant-Friedrichs-Lewy number $CLF$.

trend over the twelve probes is computed fairly well, i.e. decreasing from the cusp of the wedge to its knuckle; however, analyzing parameter variations effects on each probe separately no clear trends can be identified. Figure 13 is used to further check on this unstable behavior found on the measured (local) variables; three runs of the same SPH configuration have been performed and pressure time histories at the probe n.3 have been recorded at two different distances from the wedge face, namely at $d/h = 1.5$ and $d/h = 2.0$. For both $d/h$ values the three records show different peaks of the pressures and even slightly different time series. This behavior of the SPH solver in predicting very local variables should be further analyzed to better understand the reasons and eventually to provide a quantification of the related uncertainty.

4 URANS AND SPH SOLUTIONS COMPARISON

The two numerical solutions have been finally compared in the case of the experimental drop test performed by [17]. Results in terms of wedge vertical displacement $z(t)$ and drop speed $V_Z(t)$ are shown in Figure 14a while vertical force comparison is shown in Figure 14b. A very good agreement of both numerical solutions with the measured values of displacement and drop speed is found, confirming that the dynamics of this class of problems is well captured consistently with the results shown in the sensitivity analysis with respect to the global forces. This general agreement is also revealed by the qualitative comparison of the free surface of the two methods shown in Figure 15 at three time steps during the drop; the position of the pile-up
Figure 9: Peak pressures (left) and global vertical forces on the wedge (right) with respect to variations of the smoothing length coefficient $C_S$.

Figure 10: Peak pressures (left) and global vertical forces on the wedge (right) with respect to variations of the particles resolution $d_P$.

Figure 11: Peak pressures (left) and global vertical forces on the wedge (right) with respect to variations of the artificial viscosity coefficient $\alpha$. 
Figure 12: Peak pressures (left) and global vertical forces on the wedge (right) with respect to variations of the sound coefficient $C_{\text{Sound}}$.

Figure 13: Pressure time-history for the Probe n.3. Experimental measurements shown by black dots. Three runs of the same SPH configuration are shown by red, yellow and violet curves. Pressure probes placed at distance over smoothing length ratios of $d/h = 1.5$ (left) and of $d/h = 2.0$ (right) from the wedge border.
is well captured by both solutions while the water jet appears to be more unstable in the SPH compared to the URANS one. The vertical force agreement is again satisfactory: both the force peak and the global trend are consistent between the URANS and the SPH solutions. This final comparison, although limited to a single test, provide a basis for the practical use of both methods for the numerical prediction of such a class of hydrodynamics problems.

Figure 14: Wedge drop test (free falling), \( h_{\text{Drop}} = 1.3\, m \), \( M_{\text{Wedge}} = 94\, kg \). Comparison of the URANS and SPH solutions. Figure 14a: wedge displacement (solid curves) and wedge vertical velocity (dashed curves). Experimental results in green, URANS results in red and SPH results in blue. Figure 14b: wedge total vertical force. URANS results are shown by solid red curve; SPH results are shown by blue dots.

Figure 15: Comparison of free surface elevation at three times during the simulations. RANS on the upper row. SPH on the lower row.

5 CONCLUSIONS

The problem of numerical modeling of 2D wedge impact hydrodynamics by mesh-based and mesh-less approaches has been addressed in this study, focusing on the effect of systematic variations of the simulation set-up on the results. Open source solvers have been used for both numerical techniques: OpenFOAM libraries have been selected to achieve the RANS solution while DualSHPysics has been chosen as SPH solver. Both global and local responses have been analyzed, i.e. the total vertical force on the wedge and the peak pressures on twelve measurement probes along the side of the edge respectively. Numerical predictions have been compared to
L. Bonfiglio, S. Gaggero, A. Papetti, G. Vernengo and D. Villa

experimental measurements on a $25^\circ$ wedge for which both the displacement and the pressures time histories were available. Analyzing the results two conclusions valid for both approaches can be derived: (a) the prediction of the global forces is generally stable, i.e. not really affected by model parameter variations, consequently also the prediction of the vertical motion, and (b) the prediction of the peak pressure during the impact strongly depends on the simulation set-up. Considering the mesh-based RANS technique it is possible to find a clear trend of the local responses (the pressures) when model parameters are changed; the convergence is found but at the cost of increasing the computational effort. The sensitivity analysis on the SPH solution on the other hand reveals that the evaluation of the pressure peaks is not as robust as the prediction of the global vertical forces. The free falling drop test however shows that both solutions provide a reasonably good agreement compared to the experimental measures and are consistent in terms of force prediction.

REFERENCES


THE EFFECT OF WAVE IN-DECK IN CONVENTIONAL PUSHOVER ANALYSIS

N.U. AZMAN*, M.K. ABU HUSAINa, N.I. MOHD ZAKIb AND E. MAT SOOMc

*Technip Miri Office, 98000 Miri, Sarawak, Malaysia
e-mail: nurula@technip.com

aUTM Razak School of Engineering and Advanced Technology, Universiti Teknologi Malaysia, Jalan Semarak, 54100 Kuala Lumpur, Malaysia
e-mail: mohdkhairi.kl@utm.my

bUTM Razak School of Engineering and Advanced Technology, Universiti Teknologi Malaysia, Jalan Semarak, 54100 Kuala Lumpur, Malaysia
e-mail: noorirza.kl@utm.my

cSarawak Shell Berhad, 98100 Miri, Sarawak, Malaysia
e-mail: Ezanizam.MatSoom@shell.com

Key words: Wave-in-Deck, Wave-In-Deck Load, Probabilistic Model, Reserve Strength Ratio.

Abstract. Subsidence is not a local settlement and one of the phenomena that may be experiencing by the offshore platform throughout the platform life. Compaction of the reservoir can cause it due to pressure reduction resulted to vertical movement of soils from the reservoir to mudline. The impact of subsidence on platforms will lead to a gradually reduces wave crest to deck air gap (insufficient air gap) and causing the Wave-in-Deck (WID) on platform deck. The WID load can cause a major consequence damage to the deck structures and potential to the collapse of the entire platform. The aim of this study is to investigate the impact of WID (with and without load) on structure response for fixed offshore structure. The usual run of pushover analysis only considering the base 100-years design crest height for the ultimate collapse. Thus, by calculating the wave height at collapse using a limit state equation for probabilistic model can give a significant result for WID. It is crucial to ensure that the Reserve Strength Ratio (RSR) is not overly estimated hence giving a false impression of the value. This study is performed in order to quantify the WID load effect on producing the new revised RSR. Finally, a parametric study on the probability of failure (POF) of the platform will be performed. As part of the analysis, the USFOS Software (Non-linear) and wave-in-deck calculation as suggested by ISO 19902 as practice in the industry are used in order to complete the study. It is expected that the new revised RSR with the inclusion of WID load will be lower hence increases the POF of the platform. The accuracy and effectiveness of this method will assist the industry, especially operators, for the purpose of decision-making and, more specifically, for their outlining of action items as part of their business risk management.
1 INTRODUCTION

In Malaysia, the offshore oil and gas industry is more than 100 years old. Its youthful economic exuberance has now given away to middle-aged restraint as the price of oil has fallen and field-development and operating costs have risen. In finding ways of managing the various financial risks, together with hydrocarbon exploration and production at sea, the structural reliability assessment has introduced, i.e., a rational method of putting the economics and engineering of offshore structures into a context that takes due account of uncertainties, particularly those connected with severe ocean storms [1].

Offshore jacket platforms are commonly used in the oil and gas production in the shallow water depths of Malaysia. Over 250 installations have been operating for more than 20 years [2]. 48% of these platforms have already exceeded 25 years reaching their initial design life of 20 to 25 years [3]. In view of the continuous production required beyond the design life, life extension of these installations is inevitable.

Development of the energy sector specifically in oil and gas with resources becoming scarce and challenging, added with growing development cost, has demanded oil and gas companies to enhance the recovery of oil and gas resources from developed fields and/or develop new discovery reserves from existing oil and/or gas platforms. In some cases, with several contributing success factors, this approach has proven to give a significant reduction in development costs, resulting in good project economics, making it viable to recover more oil and gas resources [4].

This paper is composed of 6 sections. Section 1 presents the introduction of oil and gas scene in Malaysia, followed by a brief review of wave-in-deck related to offshore platforms in Section 2. Section 3 described the methodology of calculating the reserve strength ratio of the platform and followed by the test structure specification in Section 4. The comparison of results is presented in Section 5 and lastly the conclusion and recommendation based on presented results in Section 6.

2 WAVE-IN-DECK AND RESERVE STRENGTH RATIO REVIEW

Wave-in-deck (WID) occurs when there is no deck clearance or air gap between the water level and bottom steel of topsides structure when it hit by the waves. Hence it should be considered in the analysis to avoid underestimate of the extreme tether tension. [5] All offshore platform which having WID need to be adequately designed by including the WID loads, especially the topside framing and the equipment seated on the deck. Furthermore, failure of considering the WID might lead to a collapse of the platform [6].

Pushover analysis, also known as ultimate strength analysis is widely used in determining the reserve strength ratio (RSR) of an offshore platform. The platform ability to withstand a specific environmental load will be checked, i.e. 100-years environmental loads, especially for an ageing platform. In view of complexity analysis, WID effect has been excluded in the conventional pushover analysis by only considering the 100-years crest height [7]. Figure 1
shows in view of the comparison between two (2) different approaches in industry, (b) where the impact of WID happens at the level of cellar deck of the platform.

The RSR of the platform is related to the physical wave height. It means that the associated wave height can be large enough to reach the deck structure. However, the conventional method involves the increment of the 100-years environmental load without considering the changing of the wave height [8]. The USFOS software that has been widely adopted for pushover analysis consider the wave forces up to true sea surface. The wave load is scaled up proportionally but not the wave height [9].

In order to summarize the sequence of event, Bow-Tie chart is introduced in this paper. The Bow-Tie is one of Health Safety Security Environment (HSSE) tool support for As Low As Reasonably Practicable (ALARP) [10]. As part of the Bow-Tie, structure strength and RSR determination are important elements of control barrier in order to avoid for the platform submergence (Top event). Figure 2 shows the elements in Bow-Tie:
3 METHODOLOGY

In practice, the linear and non-linear analyses are implemented in this study with the base case and modified case. Begun with the base case run approach, in linear analysis with applying the original data from metocean and resulted in non-linear BS and RSR values at collapse. The HRSR level is calculated from the limit state equation for probabilistic model as per Eq. (1);

\[ \frac{H_{\text{RSR}}}{H_{100}} = \frac{\text{RSR}_1}{\alpha} \]  

Basically, this study only considers wave in deck in horizontal directional impact. The modified case run approach is estimated wave crest height at cellar deck for re-linear analysis prior to re-non-linear analysis. As part of the analysis, the Silhouette (ISO 19902) approach has considered for representing the wave/current action load on topside, \( E_{\text{topside}} \) as defined below

\[ E_{\text{topside}} = \frac{1}{2} \rho_w C_a (\alpha_{wk} U_w + \alpha_{cb} U_c)^2 A \]  

where

- \( \rho_w \) = is the density of sea water
- \( U_w \) = is the fluid velocity
- \( U_c \) = is the current speed in line with the wave
- \( \alpha_{wk} \) = is the wave kinematic factor (0.88 for tropical cyclones and 1.0 for winter storm.
- \( \alpha_{cb} \) = is the current blockage factor for the structure

In preparation of the non-linear pushover analysis, the latest analysis SACS model has used analysis. The method of calculating the RSR of the platform is presented in the flowchart below:
Figure 3: Flowchart of Analysis Procedure

(A) SACS Model Preparation:
   i) Verify the model against as-built drawings and weight control report
   ii) Verify the metocean data for 100-years return period, i.e. maximum wave height, $H_{100}$, associated period, $T_{ass}$
   iii) Perform long term distribution for item iii)

(B) USFOS Model Preparation [11]:
   i) Convert the SACS model to UFO format: model.fem
   ii) Prepare USFOS header file: header.fem

(C) Non-Linear Pushover Analysis:
   i) Run pushover analysis until platform is collapsed
   ii) Determine the base shear and RSR at collapse
   iii) Determine the failure mode

(D) Air Gap Analysis:
   i) Determine the wave height at collapse, $H_{RSR}$ using formula introduced by [12]:
      $H_{RSR}/H_{100} = RSR^{1/\alpha}$
   ii) Check the wave crest at $H_{RSR}$ against the cellar deck elevation, $CD_{EL}$

(E) Wave-In-Deck Load, WID:
   i) Calculate WID load using formula in ISO 19902 [13] and include it in the SACS model
   ii) Repeat step (B) and (C)

(F) Reserve Strength Ratio, RSR and Probability of Failure, POF
   $RSR = \frac{\text{Base Shear at Collapse}}{\text{Base Shear 100-years}}$
   POF calculation using Reliability Based Design and Assessment method [14]
4 TEST STRUCTURE SPECIFICATION

The test structure is an ageing compression fixed template platform with a water depth of 88.9m during installation. The general outline of the platform is shown in Table 1 and Figure 4. This platform has been selected due to the subsidence event that takes place throughout the platform life. Based on the subsidence report, there is a potential of WID occurrence due to a high level of subsidence at the area.

Table 1: Platform Specification

<table>
<thead>
<tr>
<th>Features</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Field</td>
<td>Sarawak (Malaysia)</td>
</tr>
<tr>
<td>Design Service Category</td>
<td>Compression</td>
</tr>
<tr>
<td>Design Safety Category</td>
<td>Manned</td>
</tr>
<tr>
<td>Installed</td>
<td>1999 (18 years)</td>
</tr>
<tr>
<td>Water Depth During Installation</td>
<td>88.90m</td>
</tr>
<tr>
<td>Projected Water Depth at the year 2024</td>
<td>Minimum: 94.36m, Maximum: 95.79m</td>
</tr>
<tr>
<td>Platform Orientation</td>
<td>Platform North, PN is orientated at 30° (clockwise) relative to True North, TN</td>
</tr>
<tr>
<td>Deck Configuration</td>
<td>MSF Deck (+23.40m) &amp; Cellar Deck, CD (+15.80m)</td>
</tr>
<tr>
<td>Platform Brace Type</td>
<td>Vertical diagonal brace</td>
</tr>
<tr>
<td>Leg</td>
<td>4 with diameter of 1485.9mm and 19.05mm thickness, grouted annulus</td>
</tr>
<tr>
<td>Number of Pile</td>
<td>4 with diameter of 1371.6mm and 3.81mm thickness, 109.8 m penetration below mudline</td>
</tr>
<tr>
<td>Number of Caisson</td>
<td>2</td>
</tr>
<tr>
<td>Bridge Link</td>
<td>2, located at platform West and East</td>
</tr>
</tbody>
</table>

The platform is modelled and verified using Structural Analysis Computer Software (SACS) computer program. The pile-soil-interaction, PSI was also modelled as it would also affect the RSR of the platform [15] in term of P-y, T-z and Q-z. Afterwards, the non-linear pushover analysis will be performed using USFOS computer program as described in Section 3.
5 COMPARISON OF THE RESULT FOR WITHOUT AND WITH INCLUSION WAVE-IN-DECK

In this section, the result between the methods is tabulated in detail. Omnidirectional metocean data is selected for the pushover analysis. Eight (8) direction corresponding to 0°, 47°, 90°, 132°, 180°, 227°, 270° and 312° were performed to the test structure. It is derived from the result of Metocean data analysis and was provided in this case study. The Metocean data was derived using existing HINDCAST data and it is based on deep water hydrodynamic [14].

5.1 METOCEAN DATA AND ASSESSMENT WATER DEPTH

Metocean data and water depth for 100-years return period are tabulated in Table 2 below. The sensitivity of minimum and maximum water depth need to be performed beforehand.

Table 2: Metocean Data and Water Depth for 100-Years Return Period

<table>
<thead>
<tr>
<th>Wave Height (m)</th>
<th>Wind Speed at 10m from MSL (m/s)</th>
<th>V (1-hr)</th>
<th>V (1-min)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hmax</td>
<td>Wind Speed</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Wave Period (s)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tass (lower)</td>
<td>10.1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tass (Central)</td>
<td>11.3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tass (Upper)</td>
<td>12.4</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Associated Current (m/s)</th>
<th>Minimum</th>
<th>Maximum</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.00d, Surface</td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.90d</td>
<td>0.900</td>
<td></td>
</tr>
<tr>
<td>0.80d</td>
<td>0.828</td>
<td></td>
</tr>
<tr>
<td>0.70d</td>
<td>0.756</td>
<td></td>
</tr>
<tr>
<td>0.60d</td>
<td>0.684</td>
<td></td>
</tr>
<tr>
<td>0.50d</td>
<td>0.612</td>
<td></td>
</tr>
<tr>
<td>0.40d</td>
<td>0.540</td>
<td></td>
</tr>
<tr>
<td>0.30d</td>
<td>0.475</td>
<td></td>
</tr>
<tr>
<td>0.20d</td>
<td>0.456</td>
<td></td>
</tr>
<tr>
<td>0.10d</td>
<td>0.430</td>
<td></td>
</tr>
<tr>
<td>0.06d</td>
<td>0.389</td>
<td></td>
</tr>
<tr>
<td>0.04d</td>
<td>0.351</td>
<td></td>
</tr>
<tr>
<td>0.02d</td>
<td>0.243</td>
<td></td>
</tr>
<tr>
<td>0.00, Seabed</td>
<td>0.118</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Water Level (m)</th>
<th>Minimum</th>
<th>Maximum</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean Sea Level, MSL at Year 2024</td>
<td>94.36</td>
<td>95.79</td>
</tr>
<tr>
<td>Highest Astronomical Tide, HAT</td>
<td>-</td>
<td>0.9</td>
</tr>
<tr>
<td>Lowest Astronomical Tide, LAT</td>
<td>-1.2</td>
<td>-</td>
</tr>
<tr>
<td>Surge</td>
<td>-</td>
<td>0.6</td>
</tr>
<tr>
<td>Inaccuracies of Water Depth</td>
<td>-0.94</td>
<td>0.95</td>
</tr>
<tr>
<td>Assessment Water Depth</td>
<td>92.22</td>
<td>98.24</td>
</tr>
</tbody>
</table>

5.2 RSR AND H_{RSR} DETERMINATION

The outcome from the pushover run using USFOS Software, Table 3 below shows the result of each omnidirectional without the inclusion of wave-in-deck. It is conventional approach with the normal process of determination of RSR, H_{RSR} and Wave Crest.
Table 3: RSR, H_{RSR} and POF without Inclusion of Wave-In-Deck

<table>
<thead>
<tr>
<th>Run No.</th>
<th>Dir. (°)</th>
<th>RSR</th>
<th>Base Shear at Collapse (kN)</th>
<th>H_{RSR} (m)</th>
<th>Wave Crest at H_{RSR} (m)</th>
<th>Probability of Failure, POF</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>5.58</td>
<td>24217.31</td>
<td>31.89</td>
<td>111.93</td>
<td>*</td>
</tr>
<tr>
<td>2</td>
<td>47</td>
<td>5.94</td>
<td>22186.96</td>
<td>33.08</td>
<td>112.79</td>
<td>*</td>
</tr>
<tr>
<td>3</td>
<td>90</td>
<td>5.92</td>
<td>20365.43</td>
<td>33.02</td>
<td>112.77</td>
<td>*</td>
</tr>
<tr>
<td>4</td>
<td>132</td>
<td>6.14</td>
<td>22645.99</td>
<td>33.74</td>
<td>113.28</td>
<td>*</td>
</tr>
<tr>
<td>5</td>
<td>180</td>
<td>4.74</td>
<td>21140.49</td>
<td>28.97</td>
<td>109.85</td>
<td>3.65E-04</td>
</tr>
<tr>
<td>6</td>
<td>227</td>
<td>5.87</td>
<td>22179.10</td>
<td>32.85</td>
<td>112.64</td>
<td>*</td>
</tr>
<tr>
<td>7</td>
<td>270</td>
<td>6.44</td>
<td>22553.59</td>
<td>34.70</td>
<td>114.00</td>
<td>*</td>
</tr>
<tr>
<td>8</td>
<td>312</td>
<td>6.86</td>
<td>24850.19</td>
<td>36.01</td>
<td>114.94</td>
<td>*</td>
</tr>
</tbody>
</table>

* Only the POF of the lowest RSR is calculated.

The distance of the cellar deck bottom of steel, BOS from the mudline is 103.79m, hence all cases having wave crest at H_{RSR} higher than the BOS. Lowest RSR which is run no. 5 (at 180° omni directional) is selected for further analysis by considering the wave-in-deck load. Refer Table 4 below for revised RSR and POF:

Table 4: RSR and POF with Inclusion of Wave-In-Deck

<table>
<thead>
<tr>
<th>Run No.</th>
<th>Dir. (°)</th>
<th>RSR_{WID}</th>
<th>Base Shear at Collapse (kN)</th>
<th>Probability of Failure, POF_{WID}</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>180</td>
<td>4.54</td>
<td>20234.25</td>
<td>4.54E-4</td>
<td>RSR_{WID} &lt; RSR_{WID} &gt; POF</td>
</tr>
</tbody>
</table>

6 CONCLUSION

From the result above, it can be concluded that an assessment is completed when satisfactory compliance with respect to RSR_{WID} is less than RSR and following conclusions can be drawn:

- The RSR_{with WID} is lower compared with RSR_{without WID} i.e. 4.54 < 4.89 while POF_{with WID} is higher compared with POF_{without WID} i.e. 4.54E-04 > 3.65E-04. Thus the result is acceptable.
- It is crucial to consider the wave-in-deck loads in the pushover analysis hence to avoid overestimation in the value.
- Even though that the wave-in-deck load was difficult to predict, it cannot be ignored totally hence giving a false impression and will lead to a wrong judgement in later assessment.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the support of the academic and industrial establishments they present. This research has been undertaken by leading author while working for Technip Miri Office, Miri, Sarawak, Malaysia and would like to express her gratitude to co-author from Sarawak Shell Berhad for the data and advice support. The paper is
financially supported by the Universiti Teknologi Malaysia (Malaysia) [grant number: Q.K130000.2540.11H30 / Q.K130000.2540.12H04] which is gratefully acknowledged.

REFERENCES

Trim optimisation in waves

AURELIEN DROUET*, PIERRICK SERGENT*, DORIANE CAUSEUR* AND PHILIPPE CORRIGNAN†

* HydrOcean
8 boulevard A. Einstein, Nantes, France
E-mail: contact@hydrocean.fr, web page: http://www.hydrocean.eu

† Bureau Veritas Marine & Offshore
8 boulevard A. Einstein, Nantes, France
E-mail: philippe.corrignan@bureauveritas.com, web page: http://www.bureauveritas.fr

Key words: Trim optimisation, Added resistance in waves, RANSE, VOF

Summary: The trim optimisation is nowadays a common practice for all ship owners. Reduction of fuel consumption and improvement of ship energy efficiency concern all sectors of the maritime industry. The trim optimisation consists in finding the best trim angle with regards to the lowest power - i.e. fuel consumption - for a given operating condition (loading condition and speed). Although the operational efficiency of trim optimisation based on calm water resistance computations databases could be proven at sea, it is well known that added resistance due to wave can significantly impact fuel consumption. The increase of cluster computing power make it today possible to evaluate the added resistance in waves using state of the art free surface RANSE solvers. This study of the optimal trim depending on the sea-states show that taking into account the waves could have a significant impact on the optimal trim that have been identify on still water. Although the optimal trim angle trends are the same on still water and in waves for low sea-states, for highest sea-states the trends can be the perfect opposite. This study shows that the optimal trim mainly depends on the draft and the sea-state. The speed seems to have an effect on the gains / losses amplitude but not on the best trim value. The in waves databases will enable operators to predict their ship efficiency with better accuracy, taking into account weather predictions. The trim needs to be therefore adapted regularly depending on the weather conditions the ship encounter during her voyage.

1. INTRODUCTION

The trim optimisation is nowadays a common practice for all ship owners. Reduction of fuel consumption and improvement of ship energy efficiency concern all sectors of the maritime industry. Even though the oil price is low, ship owners are still asking for more economical ships. Regulation now also imposes that ship owners reduce air emissions (NOX, SOX) of their ships. The trim optimisation is considered as one of the most easily achievable and cost effective fuel saving practices, as it requires no ship modification. The trim optimisation consists in finding the best trim angle with regards to the lowest power - i.e. fuel consumption - corresponding to a given operating condition (loading condition and speed). The ship trim is then set by changing the location of the centre of gravity issued from the deadweight weight distribution or issued from ballasting. The trim optimisation databases provided by HydrOcean
are generally generated from 100 free surface RANSE computations combining speed, displacement and longitudinal position of the centre of gravity (directly linked to the static trim). They are routinely based on calm water resistance simulations for CPU cost effectiveness.

HydrOcean has developed and distributes a dedicated trim optimisation Database and Software (figure 1) that enables a ship’s crew to determine the optimal trim at given ship displacements and speeds. It has shown its efficiency in terms of return of investment. As an example, for a container ship with a consumption of 25 000t / year, about 1.0 % of Heavy Fuel Oil (250t / Year / Vessel) are saved thanks to the trim optimisation.

Although the operational efficiency of trim optimisation based on calm water resistance computation databases could be proven at sea, it is well known that added resistance due to wave can significantly impact fuel consumption. The increase of cluster computing power make it possible today to evaluate the added resistance in waves using state of the art free surface RANSE solvers.

This paper presents a comparison between a trim optimisation carried out on calm water with one performed in waves. The methodology and the numerical parameters are firstly presented and then the results and conclusions.

2. METHODOLOGY

This part presents the methodology used in order to compute the mean drift forces according to a given irregular sea-state. The mean drift forces could be evaluated using direct computations on an irregular sea-state or using post-processing based on added resistance transfer function called further ARTF. Although the first solution is more accurate, the requested CPU time are too long for this kind of application since the mean drift forces need to be evaluated for a large amount of sea states which require one computation by sea-state. The second solution was therefore chosen since the computation of ARTF enable to determine the mean drift forces for many sea-states just by post-processing and without requiring new CPU time. This second methodology is described hereafter.
The estimation of mean drift forces is divided into 4 main steps:
1. Modeling the ship with a forward speed for various frequencies in 1m regular waves (CFD computations).
2. Building an added resistance transfer function issued from step 1 results.
3. Combination of an added resistance transfer function and a wave spectrum
4. Determination of the mean drift force in irregular waves for a given sea-state.

Please note that for the second point, the ARTF depends on the ship speed and displacement.

The estimation of added resistance in regular waves is issued from ITTC recommendation [6]. The mean added resistance in waves is obtained by the subtraction of the still water resistance from the total resistance in waves (equation 1 and figure 2).

\[
R_{AW} = \overline{R_T} - R_{SW}
\]
\[
R_{AW} : \text{Mean added wave resistance}
\]
\[
R_{SW} : \text{Still water resistance}
\]
\[
R_T : \text{Total resistance in waves}
\]

The estimation of the added resistance in regular waves therefore requires a preliminary still water resistance computation.

The mean added resistance in waves is computed for 1 m regular waves for a range of wave length (generally from 0.2Lpp to 2.0) that enables to reproduce all physical phenomena (diffraction, radiation, motion resonance …) that appears depending on the wave length. The added resistance transfer function is then computed by the following dimensionless equation.
\[
\text{ARTF} = \frac{R_{AW}}{\rho g A^2 B^2 / L_{pp}}
\]

With:
- \(R_{AW}\): Mean added wave resistance
- \(\rho\): Water density
- \(A\): Wave amplitude
- \(B\): Ship beam
- \(L_{pp}\): Ship length between perpendicular

Although the wave spectrum chosen for the whole study is Pierson-Moskowitz other kinds of spectra such as Jonswap or Bretschneider-Mitsuyau could have been used for this methodology. This choice was made according to the typical commercial route of the container ship. Depending on the weather condition the ship will encounter during his voyage, the wave spectrum parameters will be adapted with regards to \(H_s, T_p\).

The Pierson-Moskowitz wave spectrum is given by the following equation [5].

\[
S(\omega) = 5\pi^4 \frac{H_s^2}{T_p^4} \cdot \frac{1}{\omega^5} \exp\left[-\frac{20\pi^4}{T_p^4} \cdot \frac{1}{\omega^4}\right]
\]

With:
- \(H_s\): Significant wave height
- \(T_p\): Peak wave period
- \(\omega\): Wave pulsation

An example of Pierson-Moskowitz wave spectrum with \(H_s=13\text{m}\) and \(T_p=14\text{s}\) is presented on the following graph.
Once the ARTF and the wave spectrum are defined, the last step is the mean drift forces computation using the following formulae.

$$\overline{R_{AW}} = 2 \int_0^{\infty} \frac{R_{AW}}{A^2} S(\omega) d\omega$$

With:
\[ R_{AW} : \text{Mean added wave resistance} \]
\[ A : \text{Mean added wave amplitude used to compute the ARTF} \]
\[ S(\omega) : \text{Wave spectrum} \]  

(4)

3. NUMERICAL PARAMETER
This part presents the numerical parameters of the CFD computations.

3.1. Coordinate systems
The coordinate system is described below:
- Origin is located at the intersection of symmetry plane, rudder axis, keel line
- X axis is oriented toward ship bow
- Y axis is oriented port side
- Z axis is oriented up
3.2. Geometry

The geometry of the container ship used for the trim optimisation study is presented on the figure below. For confidentiality reasons, the hydrostatic particulars and main dimensions are not provided.

![Figure 6: Container ship geometry](image)

3.3. Simulation parameters

The simulation were carried out with ISIS-CFD solver (described in the next paragraph) at full scale using the k-ω SST (Menter) turbulence model. A fixed velocity is imposed to the hull from rest to the target speed. Simulations are unsteady, with free heave and pitch. Water characteristics are for salt water (15°C) issued from ITTC [9] (Density 1026.0210 kg/m³, Dynamic viscosity: 0.001220 Pa.s). LCG is calculated to reach the required static equilibrium position corresponding to each trim condition. VCG is assumed to be at free surface. The displacement is fixed when varying the trim.

3.4. Solver description

ISIS-CFD flow solver developed by ECN (Ecole Centrale de Nantes) uses the incompressible unsteady RANSE (Reynolds Averaged Navier-Stokes Equations) as the governing equations. The equations of k-ω SST turbulence model are used for the closure of RANSE.

ISIS-CFD solver is based on the finite volume method to build the spatial discretization of the transport equations. The velocity field is obtained from the momentum conservation equations and the pressure field is extracted from the mass conservation constraint, or continuity equation, transformed into a pressure equation. In the case of turbulent flows, additional transport equations for modelled variables are solved in a form similar to those of the momentum equations and they can be discretized and solved using the same principles. Incompressible and non-miscible flow phases are modelled through the use of conservation equations for each volume fraction of phase [1].

ISIS-CFD solver has been validated through lots of benchmark cases by comparing with towing tank model test. It ensures the accurate estimation of ship hull hydrodynamic performance. It is also possible to propagate regular waves from boundary conditions.
3.5. Matrix of computations

In order to be able to compare the trim optimisation in waves with the trim optimisation in still water the scope of work is divided into two parts corresponding respectively to still water resistance computations and added wave resistance computations. The trim optimisation was carried out with regards to 2 drafts and 2 speeds. For each operating condition 4 trims were evaluated. 7 wave periods will be computed in order to build the ARTF corresponding to each condition. To sum up, 16 still water resistance and 112 added resistance in waves simulations were performed.

3.6. Grid characteristics

This part presents the grid characteristics of the calm water grids and then the special parameters of the grids used for added resistance in waves simulations. The calm water grids are unstructured and composed of 3.5 million cells per grid. 16 grid was generated, one per combination of draft and trim. Wall function were used with $30 < Y+ < 300$ corresponding to ITTC recommendation [7,8]. Some grid views for the 11.0 m draft even keel are presented hereafter.

![Figure 7: Grid views for the 11.0 m draft even keel](image)

Some additional refinement are used into the grids used for added resistance in waves simulations because of the wave propagation. The main criteria to be respected concern the ratio between the wave amplitude with the vertical cell size where $H/dz \geq 8$ and the radio between the horizontal cell size with the vertical cell size where $d_x/dz < 8$. The grid is therefore finer for wave propagation with about 5 million cells per grid.

![Figure 8: Comparison between resistance grid refinement (left) and in waves grid refinement (right).](image)
4. RESULTS

This part presents firstly the analysis of the calm water resistance results computed for different combination of speed, draft and trim. The analysis of the added resistance in waves results with regards to the optimal trim on the same operating condition as the still water resistance is then presented and compared to the still water results.

4.1. Still water resistance

The tables below show the differences in terms of resistance with regards to the even keel configuration for four operation conditions issued from the combination of speeds drafts.

Table 1: Differences in terms of ship resistance with regards to the even keel configuration, 4 operating conditions

<table>
<thead>
<tr>
<th>T (m)</th>
<th>Trim (m)</th>
<th>V (kn)</th>
<th>Diff Rts (%) / Trim 0</th>
</tr>
</thead>
<tbody>
<tr>
<td>11.0</td>
<td>-2.0</td>
<td>15.0</td>
<td>1.03%</td>
</tr>
<tr>
<td>11.0</td>
<td>-1.0</td>
<td>15.0</td>
<td>0.91%</td>
</tr>
<tr>
<td>11.0</td>
<td>0.0</td>
<td>15.0</td>
<td>0.00%</td>
</tr>
<tr>
<td>11.0</td>
<td>1.0</td>
<td>15.0</td>
<td>-1.14%</td>
</tr>
<tr>
<td>14.0</td>
<td>-2.0</td>
<td>18.0</td>
<td>3.19%</td>
</tr>
<tr>
<td>14.0</td>
<td>-1.0</td>
<td>18.0</td>
<td>0.40%</td>
</tr>
<tr>
<td>14.0</td>
<td>0.0</td>
<td>18.0</td>
<td>0.00%</td>
</tr>
<tr>
<td>14.0</td>
<td>1.0</td>
<td>18.0</td>
<td>-0.34%</td>
</tr>
</tbody>
</table>

Although the trend is the same (bow down is better), the amplitude of gains / losses is different between the operating conditions with gains varying from 0.2% to 2.76% and losses varying from 2.77% to 4.22% for the same trim range. Depending on the trim configuration, the bow and stern contributions to the resistance are different and therefore lead to gains or losses with regards to the even keel configuration.

The figures below illustrate the differences in terms of interactions between the water surface and the bulbous bow when changing the trim.

Figure 9: Interactions between the water surface and the bulbous bow different when changing the trim

These interactions are among the origins of the trim optimisation gains or losses. The second main factor is the stern immersion, as shown on the figures below. There are three main
phenomena that contribute to the ship resistance around the stern: the stern slope, the wetted length, and the transom immersion.

Figure 10: Differences in terms of stern slope, the wetted length when changing the trim

Figure 11: Differences in terms of transom immersion / transom wetness when changing the trim

4.2. Added resistance in waves

The graphs below show the ARTF computed for the same four operating conditions and four trims simulated in still water resistance. The range of wave length is between 0.2Lpp and 1.6Lpp and distributed according to 7 values chosen to well describe the different parts of the curve (3 around the peak, 2 around the flat part corresponding to the short periods and 2 in the slope between the peak and the large periods). As expected, all ARTF have the same shape. A horizontal flat shape is obtained for short periods below 0.2Lpp. In this part, the motion are very low and the main contribution of forces is due to the wave diffraction generally called drift forces. The peak of added resistance in waves is mainly due to motion radiation. The peak frequency is very close to the ship length which corresponds to the motion resonance frequency of the pitch motion. As far as the wave length increases (to the right of the curve) the wave conditions are closer to the still water and therefore the added resistance in waves is decreasing to zero.
ARTF curves show that as the speed increases, the peak is higher and slightly moved to the left due to the reduction of the encounter periods. The effect of the trim is more visible for the draft 11.0 m than for the draft 14.0 m. This is mainly due to the fact that the bulbous bow is closer to the free surface for the draft 11.0 m and fully immersed for the draft 14.0 m. The figures below illustrate the bulbous bow immersion in waves for the drafts 11.0 m and 14.0 m.

The trim optimisation in waves depends of the sea-state the ship will encounter during his voyage. In order to have a look on a large range of sea-states, the mean drift forces - computed according to the methodology presented in the paragraph 2 – were estimated for a full matrix composed of 14 peak wave periods from 4.0 s to 17.0 s and 16 significant wave heights from 0.0 m to 15.0 m. The mean drift forces were evaluated also for the 4 trims. A database composed of 896 values of total resistance in waves was therefore determined as post-processing of the CFD computations (ARTF) enabling to study the best trim with regards to a given sea-state. The figure below show an example of the differences in terms of total resistance in waves with regards to the even keel trim configuration for the 224 sea-states. In the following example, the trim is equal to 1.0 to bow. Please note that some of the following sea-states are not realistics.
Figure 14: Example of the differences in terms of total resistance in waves with regards to the even keel trim configuration for the 224 sea-states for a trim equal to 1.0m to bow.

The figures below illustrates the gains and losses in terms of total resistance in waves with regards to the even keel trim condition for each sea-state evaluated.

The trends highly depend on the considered draft as already discussed because of the interactions between the bulbous bow and the free surface. For low drafts, the optimal trim in waves can be opposite to the one in calm water. For the low sea-states, the trends are close to the still water ones but for the high sea-states in terms of peak wave period and significant wave height, the best trim in terms of total resistance in waves is bow up which is the opposite as the still water optimal trim angle. The trend is the same for the two speeds but the amplitude of gains is larger for the higher speed. For the higher drafts, the optimal trim is less impacted, but the gains are highly linked to the sea-state.
5. CONCLUSIONS

This study of the optimal trim depending on the sea-states show that taking into account the waves could have a significant impact on the optimal trim that have been identify on still water. Although the optimal trim angle trends are the same on still water and in waves for low sea-states, for highest sea-states the trends can be the perfect opposite. This study shows that the optimal trim mainly depends on the draft and the sea-state. The speed seems to have an effect on the gains / losses amplitude but not and the best trim value. The in waves databases will enable operators to predict their ship efficiency with better accuracy, taking into account weather predictions. The trim needs to be therefore adapted regularly depending on the weather conditions the ship encounter during her voyage.

These conclusions are based on mean drift forces estimated using ARTF computed for 1.0 regular wave amplitude using the assumption of a linear dependency of the added resistance in waves with the square value of the wave amplitude.

6. ACKNOWLEDGEMENTS

This work was carried within the framework of the Bureau Veritas Marine & Offshore Digital Initiatives project. The computations were performed on the IFREMER cluster CAPARMOR and on the cluster LIGER of the Institut de Calcul Intensif of the Ecole Centrale de Nantes.

7. REFERENCES

ACCELERATED FREE-SURFACE FLOW SIMULATIONS
WITH INTERACTIVELY MOVING BODIES

ARTHUR E.P. VELDMAN∗, HENK SEUBERS∗, PETER VAN DER PLAS∗
AND JOOP HELDER†

∗ Institute for Mathematics and Computer Science, University of Groningen,
P.O. Box 407, 9700 AK Groningen, The Netherlands
e-mail: {a.e.p.veldman, h.seubers, p.van.der.plas}@rug.nl

† MARIN, P.O. Box 28, 6700 AA Wageningen, The Netherlands
e-mail: j.helder@marin.nl

Key words: Computational Fluid Dynamics, fluid solid-body interaction, strong coupling, added mass, floating objects, free-fall life boat

Abstract. One of the challenges of simulating free-surface flow around moored or floating objects is the modelling, including the algorithmic coupling, of objects whose dynamics is determined by a two-way interaction with the incoming waves. The 'traditional' way of numerically coupling the flow dynamics with the dynamics of a floating object becomes unstable (or requires severe underrelaxation) when the added mass is larger than the mass of the object. To deal with this two-way interaction, a more simultaneous type of numerical coupling is being developed. The quasi-simultaneous method will be demonstrated on a number of simulation results for engineering applications from the offshore industry: the motion of a moored TLP platform in extreme waves, and the launch of a free-fall life boat.

1 INTRODUCTION

Simulating the hydrodynamics of floating structures using a two-way partitioned coupling poses a challenge when the coupling between the fluid and the structure is strong. The incompressibility of the fluid plays an important role, and leads to strong coupling when the ratio of so-called added mass to structural mass is considerate. Existing fluid-structure interaction procedures become less efficient in such cases, and can even become unstable [1, 2]. This paper proposes a coupling method that deals with the added-mass effect by anticipation, and remains stable and efficient at all times.

Physically, the interaction between incoming waves and a structure can be labelled as one-way or two-way. In the former case the structure ‘simply’ reacts to the oncoming flow
field, but in the latter case the motion of the structure influences the flow field around
the structure. The latter case poses most challenges to the numerical coupling between
flow and structure. A numerical coupling approach can be aggregated (monolithic) or
segregated (partitioned). In the former case, all discrete equations are combined into
one single set of equations. In the latter case, two separate discrete systems (modules)
can be recognized equipped with recipes to exchange information. This enhances the
flexibility of the approach, but it also requires an iterative exchange of information with
its consequences for numerical stability [3, 4]. The paper describes our efforts to find a
compromise between the robust monolithic approach and the more flexible but vulnerable
partitioned approach.

The structural model consists of a rigid body with elastic mooring lines, and the fluid
is modelled by a volume-of-fluid method for the free surface [5]. Instead of imposing
the structural displacement to the fluid, a combination of pressure and displacement is pre-
scribed that approximates the body dynamics [6]. Through this modified boundary condi-
tion, the flow can anticipate the structural motion. This anticipatory, quasi-simultaneous
coupling method increases the numerical efficiency, while the results are - by design - the
same as in the usual formulation.

To demonstrate the new method, a number of simulation results for engineering appli-
cations from the offshore industry will be presented, such as the motion of a moored TLP
platform in extreme waves, and the launch of a free-fall life boat.

\[ \mathcal{D}u = 0, \quad \frac{\partial u}{\partial t} + \mathcal{C}(u)u + \mathcal{G}p - \mathcal{V}u = f. \]
Here $\mathcal{D}$ is the divergence operator describing conservation of mass. Conservation of momentum is based on the convection operator $\mathcal{C}(\mathbf{u})\mathbf{v} \equiv \nabla (\mathbf{u} \otimes \mathbf{v})$, the pressure gradient operator $\mathcal{G} = \nabla$, the viscous diffusion operator $\mathcal{V}(\mathbf{u}) \equiv \nabla \cdot \nu \nabla \mathbf{u}$ and a forcing term $\mathbf{f}$. The kinematic viscosity is denoted by $\nu$.

Turbulence is modelled by means of large-eddy simulation (LES) using a low-dissipation QR-model as formulated by Verstappen [7]; for its use in maritime applications, see [8,9]. This model has been refined and extended in the PhD thesis of Rozema [10] towards the so-called anisotropic minimum dissipation (AMD) model, which can be applied at (highly) anisotropic computational grids. It is being further explored in cooperation with the Center for Turbulent Research (Stanford University) [11–13].

The Navier–Stokes equations (1) are discretized on an Arakawa C-grid. The second-order finite-volume discretization of the continuity equation at the ‘new’ time level $n+1$ is given by

$$
\mathcal{D}^0 \mathbf{u}^{n+1} = -\mathcal{D}^\Gamma \mathbf{u}^{n+1}_\Gamma,
$$

where $\mathcal{D}^0$ acts on the interior of the domain and $\mathcal{D}^\Gamma$ acts on the boundaries of the domain (with $\mathbf{u}_\Gamma$ denoting the velocity at the boundary). In the discretized momentum equation, convection $\mathcal{C}(\mathbf{u}_h)$ and viscous diffusion $\mathcal{V}$ are discretized explicitly in time. The pressure gradient is discretized at the new time level. In this exposition, for readability reasons the first-order forward Euler time integration will be used. In the actual simulations, a second-order Adams–Bashforth method is being applied.

Letting the diagonal matrix $\Omega$ denote the matrix containing the geometric size of the control volumes, gives the discretized momentum equation as

$$
\Omega \frac{\mathbf{u}^{n+1} - \mathbf{u}^n}{\delta t} = -\mathcal{C}(\mathbf{u}^n) \mathbf{u}^n + \mathcal{V} \mathbf{u}^n - \mathcal{G} p^{n+1} + \mathbf{f}.
$$

For divergence-free velocity fields $\mathbf{u}$, the conservative discrete convection operator is skew-symmetric. In this way, the discrete convection does not contribute to energy production or dissipation; see Verstappen and Veldman [14]. In particular, its discretization preserves the energy of the flow and does not produce any artificial diffusion. To make the discretization fully energy-preserving, the discrete gradient operator and the divergence operator must be each other’s negative transpose, i.e. $\mathcal{G} = -\mathcal{D}^{\theta T}$, thus mimicking the analytic symmetry $\nabla = -(\nabla \cdot)^T$. Then, also the work done by the pressure vanishes discretely. The latter is computed in the standard way from the Poisson equation obtained by combining (2) and (3).

The liquid region and the free liquid surface are described by an improved VOF-method; see e.g. Hirt and Nichols [15], Kleefsman et al. [5] and Düz [16].

### 3 FLUID–SOLID BODY COUPLING

A further step is to allow the moving object to interact with the fluid dynamics, e.g. it is floating on the water surface. This physical two-way coupling has to be mirrored in
the numerical coupling algorithm between the flow solver and the solid-body solver. The coupling takes place along the common interface $\Gamma$ between the fluid and the solid body; quantities involved are the dynamics (position and acceleration) of the solid body, and the dynamics of the fluid (in particular its pressure loads); see Fig. 2.

![Figure 2: Schematic partitioning between fluid and solid body, with information exchange along their common interface $\Gamma$.](image)

1. The **coupling conditions** at the interface between fluid and solid body basically express ‘continuity’ of the physics on both sides of the common interface $\Gamma$. The kinematic condition expresses that the boundary of the liquid region (partially) coincides with the surface of the solid body. In particular, material particles on both sides have the same velocity and acceleration (when the no-slip condition does not hold, only the normal velocity component is continuous). Hence, it makes sense to talk about the velocity at the interface $u_\Gamma$. Additionally, the dynamic coupling condition is based on Newton’s 3rd law “action $=$ reaction”, which expresses equilibrium of forces. In particular, we denote the fluid pressure by $p_\Gamma$. In the sequel, we will formulate the coupled problem in terms of the two interface variables: the velocity $u_\Gamma$ and the integrated (force and moment) pressure load $F(p_\Gamma)$ (short: $F_\Gamma$).

2. The **solid-body dynamics** is governed by an equation describing the acceleration $\ddot{u}$ of the solid body when reacting to the forces and moments $f$ exerted by the fluid. The latter are found by integration of the liquid pressure over the common interface $\Gamma$. In abstract terms we denote this relation by

$$M_{sb}(\dot{u}) = F_\Gamma.$$  \hspace{1cm} (4)

The mass operator $M_{sb}$ (6 DOF) describes the inertial properties of the solid body, thus the eigenvalues of $M_{sb}$ are proportional to the body mass and its moments of inertia. The load operator $F_\Gamma$ involves the integration of the fluid pressure $p_\Gamma$ over the interface to obtain the force and moments acting on the body. Note that we do not show eventual external forces, as they play no essential role in the coupling algorithm.
3. The *fluid dynamics* governs the reaction of the fluid to the motion of the solid body. The latter creates a boundary condition along the interface $\Gamma$ which has to be added to the Navier–Stokes equations (1). As a result, the pressure loads along the interface and acting on the solid body can be computed. In abstract notation we write

\[ \text{fluid dynamics:} \quad \mathcal{F}_\Gamma = -M_{\text{ad}} \ddot{u}_\Gamma, \quad (5) \]

where $M_{\text{ad}}$ is, by definition, the so-called *added mass* operator. Again, terms in $u$ have been omitted because of their minor role in the analysis of the coupling process.

The above formulation (4)+(5) in principle shows two equations for the two unknowns along the interface: loads and acceleration. Their coupling can be done in an aggregated or segregated way. An aggregated (or strong) coupling recombines both equations (or modules) into one single global system which is solved simultaneously. In contrast, a segregated (or weak) coupling keeps the two equations apart and solves them in an iterative way. The former (monolithic) approach may not always be possible, e.g. due to their ‘black-box’ character, or due to the large complexity of the modules; the latter (partitioned) approach may diverge.

### 3.1 Weak coupling

In practice, often a segregated approach is followed. The usual way is to let the fluid determine the load field (= integrated pressure forces) $\mathcal{F}^\text{old}_\Gamma$ which moves the solid body:

\[ M_{sb} \ddot{u}^\text{new}_\Gamma = \mathcal{F}^\text{old}_\Gamma. \quad (6) \]

This motion of the solid body is then transferred to the fluid, to react with a new pressure field:

\[ \mathcal{F}^\text{new}_\Gamma = -M_{\text{ad}} \ddot{u}^\text{new}_\Gamma. \quad (7) \]

In this way, we effectively have created an iterative process

\[ \mathcal{F}^\text{new}_\Gamma = -M_{\text{ad}} M_{sb}^{-1} \mathcal{F}^\text{old}_\Gamma. \quad (8) \]

The iterations in such a weak coupling method will converge if and only if the spectral radius of the iteration matrix $\rho(M_{\text{ad}} M_{sb}^{-1}) < 1$. Thus, this convergence condition is a requirement for the ratio between added mass and body mass. Roughly speaking, the solid body should be heavy enough. If it is not, underrelaxation can help achieving convergence, but this will require (many) additional (sub)iterations and diminishes efficiency.

In practice, such iterations can be implemented as follows in the solution process for the Navier–Stokes equation (3) during the time step from $n \rightarrow n + 1$. An additional subiteration process (with iteration count $k$) is included:

\[
\text{solid-body dynamics:} \quad M_{sb} (\ddot{u}_\Gamma + 1)^k = (\mathcal{F}_\Gamma + 1)^k, \quad (9)
\]

\[
\text{Navier–Stokes:} \quad (\mathcal{F}_\Gamma + 1)^{k+1} = -M_{\text{ad}} (\ddot{u}_\Gamma + 1)^k. \quad (10)
\]
If necessary, these subiterations can be made convergent by applying (severe) underrelaxation without disturbing the time accuracy of the time integration method, but often at a considerable computational price.

Below we will demonstrate with an example of a falling life boat how many subiterations can be required in practical situations. Because of this inefficiency, it is better to apply an aggregated approach (which does not require subiterations), or at least to be as close as possible to such an approach. Such an approach is the quasi-simultaneous method, originally developed for interacting aerodynamic boundary layers along airplane wings [17]; a historic overview is provided in [18, 19].

### 3.2 Quasi-simultaneous coupling

For iterative efficiency reasons, it is worthwhile to follow the monolithic, simultaneous approach as much as possible. With the full dynamics operator \( M_{sb} \) being too complex, a good approximation is sought \( \tilde{M}_{sb} \) which anticipates the reaction of the full dynamics \( M_{sb} \) and which is simple enough to be used as a boundary condition inside the Navier–Stokes solver (1). Such an approximation has been termed an *interaction law* [17–19]

\[
\tilde{M}_{sb} \ddot{u}_\Gamma = \mathcal{F}_\Gamma.
\] (11)

The interaction law (11) is then incorporated in the time integration process in defect formulation:

- **interaction law:**
  \[
  \ddot{u}_\Gamma^{n+1} - \tilde{M}_{sb}^{-1} \mathcal{F}_\Gamma^{n+1} = (M_{sb}^{-1} - \tilde{M}_{sb}^{-1}) \mathcal{F}_\Gamma^n
  \] (12)

- **Navier–Stokes:**
  \[
  \mathcal{F}_\Gamma^{n+1} + M_{ad} \ddot{u}_\Gamma^{n+1} = 0.
  \] (13)

Another interpretation in terms of time integration is that the part \( \tilde{M}_{sb} \) of the dynamics equation (4) is treated implicitly and the remaining part \( M_{sb} - \tilde{M}_{sb} \) explicitly.

![Figure 3: Weak versus quasi-simultaneous coupling.](image)

The interaction law is a boundary condition for the Navier–Stokes equations along the interface \( \Gamma \). More precisely, it will be implemented as a boundary condition for the
pressure Poisson equation. The latter is derived by first rewriting the boundary condition (12) for $u$ as

$$u^n_{\Gamma} + \delta t \tilde{M}^{-1}_{sb} F(p^n_{\Gamma}) = u^n_{\Gamma} + \delta t (M^{-1}_{sb} - \tilde{M}^{-1}_{sb}) F(p^n_{\Gamma}).$$

Then the discrete momentum equation is written in the form (2), wherein the above relation is substituted through the term $\mathcal{D}^F u$. Finally a relation is obtained featuring $F_{\Gamma}$ as a boundary condition for the Poisson equation; see also [20].

The above quasi-simultaneous integration can be analysed by eliminating $\dot{u}$ from (12)+(13), which leads to

$$(I + M_{ad} \tilde{M}^{-1}_{sb}) F^n_{\Gamma} = -M_{ad} (M^{-1}_{sb} - \tilde{M}^{-1}_{sb}) F^n_{\Gamma}. \quad (14)$$

For $\tilde{M}^{-1}_{sb} \equiv 0$, the iteration process (8) is recovered, which breaks down when $M_{ad}$ is ‘too large’. But with the term $M_{ad} \tilde{M}^{-1}_{sb}$ on the left-hand side and the difference $M^{-1}_{sb} - \tilde{M}^{-1}_{sb}$ on the right-hand side, it will be clear that this process will converge when $\tilde{M}_{sb}$ is sufficiently close to $M_{sb}$, in spite of a possibly ‘large’ $M_{ad}$.

4 EXAMPLES

To show the performance of the quasi-simultaneous approach, two test cases are presented: i) a free-fall life boat; and ii) a moored TLP platform.

4.1 Falling life boat

The first test case is a simulation of a life boat dropped into calm water. Four snapshots of the simulation are shown in Fig. 4. The dynamics of the life boat is modelled by means of a 6-DOF mechanical model.

![Figure 4: Four simulation snapshots of the drop of a life boat into calm water. The colors represent vorticity; the locally refined regions are indicated.](image-url)
The fluid flow is modelled with the Navier-Stokes equations and solved on a grid consisting of about 0.5 million active (i.e. within the fluid) grid points, with local grid refinement [21, 22] around the life boat (Fig. 4).

For physical accuracy this grid is rather coarse, but the focus in these simulations is on the numerical behaviour of the coupling process. Thus both the weak coupling procedure (8) as well as the quasi-simultaneous procedure (13) have been applied. In the latter case, the interaction law is based on the under-water part of the lifeboat (as the Poisson equation is only solved under water).

![Figure 5](image-url)

*Figure 5: Left:* The number of SOR iterations per time step for the underrelaxed weak coupling method (blue) and the anticipatory quasi-simultaneous method (green)). *Right:* The estimated added mass for the falling lifeboat as a function of time. The crossing of the free surface is clearly visible in the added mass.

The most important result concerns the amount of work that is needed per time step to achieve the coupling between solid-body dynamics and fluid flow. The weak method often requires dozens of subiterations, in each of which a Poisson equation has to be solved. This number is dependent on the amount of fluid that is moved aside by the moving body, represented by the added-mass operator $M_a$. To be fair, the later subiterations have a good initial guess so they are not as expensive as the earlier ones. Thus the amount of work is better represented by the total number of iterations, in this case SOR [23], that is needed for all Poisson solves within one time step. For other matrix solvers, the situation is relatively similar, as the number of required Poisson solves is basically independent of the solver choice.

The amount of work for a typical simulation (actually another one than presented in Fig. 4) is shown in Fig. 5(left). The close relation with the added mass becomes visible when plotting the time history of the estimated added mass in Fig. 5(right). Note that the ‘gaps’ in the curve are due to loss of figures during the estimation of the added mass. Comparison with Fig. 5(left) shows clearly that the number of iterations grows rapidly when the ‘added-mass ratio’ $\rho \left( M_{ad}M_{sb}^{-1} \right)$ grows beyond 1. In contrast, the quasi-simultaneous method requires only 1 or 2 subiterations (with additional Poisson solver), resulting in much less work per time step (Fig. 5(left)). This reduction in the
number of subiterations is highly independent of the grid size and of the chosen Poisson solver. More specifically, the number of subiterations is (to first order) only dependent on the difference between the analytic pendants of $M_{sb}$ and $\tilde{M}_{sb}$.

The added mass varies greatly over time during the impact, as the boat enters the water and a larger part of the wave has to respond. It is clear that the relaxation-based method (9)+(10) is sensitive to this ratio, as the workload increases during the entry phase. The anticipative method (12)+(13) however remains efficient regardless of the added-mass ratio, as the boundary condition inside the wave simulation predicts the boat motion. It is observed that in this application the workload is reduced by a factor around 10 for the complete simulation.

4.2 Moored TLP

The second test case is a tension-leg platform in a long-crested wave. The platform is modelled as a rigid 6-DOF body with elastic mooring lines. It can perform large but finite translations and rotations in three dimensions, but it cannot deform or change in volume. The generalization of the algorithmic coupling method to deformable bodies is presented in a paper at this year’s OMAE conference [6].

The incoming wave is a nonlinear 5th-order Stokes wave. Since the variations around the waterline are small compared to the size of the platform, the added-mass ratio is relatively constant. Even for this moderate added-mass ratio however, the anticipative method outperforms the relaxation-based method by a factor 2.5 to 3.

5 CONCLUSIONS

The ComFLOW simulation method has been designed to simulate and study extreme waves and their impact on falling, floating and moored structures. In particular, in this paper the interaction between the dynamics of a structure and the oncoming wave field
Figure 7: Workload of the simulation of a tension-leg platform in a Stokes-5 wave. Left: Variation of
the effective added-mass ratio over time. Right: The workload for the simulation corresponds to the area
under the curve.

is investigated. The efficiency of the numerical coupling algorithm is largely determined
by the added-mass ratio, which seriously affects the existing partitioned coupling schemes
based on the (sequential) exchange of loads and motions: a stronger coupling with a
higher range of added-mass ratios leads to loss of performance in these traditional coupling
schemes.

Inspired by developments in aerodynamic boundary-layer theory, an anticipatory, quasi-
simultaneous coupling scheme has been developed which intends to circumvent most of
the iterative coupling action. Application to two maritime applications, a free-fall life boat
and a moored TLP, shows the anticipatory scheme to be robust and rather insensitive to
the added-mass ratio.

Future work will focus on more general applications such as interaction of multiple,
possibly interconnected, rigid bodies (e.g. the RUG Ocean Grazer [24]).

6 ACKNOWLEDGMENTS

This work is part of the research programme Maritiem 2013 with project number 13267
which is (partly) financed by the Netherlands Organisation for Scientific Research (NWO).

REFERENCES


instabilities in sequential staggered coupling of nonlinear structures and incompressible


AN INNOVATIVE TOOL TO STUDY AND OPTIMIZE RACING YACHT APPENDAGES USING FLUID-STRUCTURE INTERACTIONS

Rémy Balze\textsuperscript{1,2}, Nedeleg Bigi\textsuperscript{3}, Kostia Roncin\textsuperscript{3}, Jean-B. Leroux\textsuperscript{3}, Alain Nème\textsuperscript{3}, Vincent Keryvin\textsuperscript{1}, Antoine Connan\textsuperscript{1,4}, Hervé Devaux\textsuperscript{1} and Denis Gléhen\textsuperscript{1},

\textsuperscript{1}GSEA Design, 1 rue Galilée, F-56270 Ploemeur, France  
e-mail: remy@hds-design.com, web page: http://www.gseadesign.com/

\textsuperscript{2}Univ. Bretagne Occidentale, FRE CNRS 3744, IRDL, F-29200 Brest, France  
e-mail: remy.balze@univ-brest.fr - Web page: http://www.irdl.fr

\textsuperscript{3}ENSTA Bretagne, FRE CNRS 3744, IRDL, F-29200 Brest, France  
e-mail: nedeleg.bigi@ensta-bretagne.org - Web page: http://www.irdl.fr

\textsuperscript{4}Univ. Bretagne Sud, FRE CNRS 3744, IRDL, F-56100 Lorient, France  
e-mail: vincent.keryvin@univ-ubs.fr - Web page: http://www.irdl.fr

Key words: Racing boats, multihull, appendage, foil, rudder, fluid-structure interactions, FSI, optimization, 3D lifting line, composite beam, finite element model

Abstract. GSEA Design developed a fluid structure method (FSI) suitable for early design stage of appendage with complex shapes dedicated to the America’s Cup flying catamarans. The aerodynamic loading and the boat weight are counteracted by the appendages and mainly the dagger-board. Consequently, the appendage structural design is very critical. Based on a 3D lifting line and a modified beam element method, the GSEA Design FSI method takes less than one minute to compute. An illustrating example on a L-shape appendage shows that the FSI results compared to a non-FSI results can be particularly different at the elbow. Thanks to the short computational time of the method, multi-objective optimizations can be performed. For instance, a second illustrating example shows the optimization of the appendage weight and stiffness.
1 INTRODUCTION

A sailing boat is a wind driven vessel in permanent equilibrium between two fluids: air, that provides the power required for its movement through sails or wingsails, and water, on which the sailing boat is based. In addition to hulls, structural parts that allow this support are appendages: rudders, foils, daggerboards. . .. The three last America’s Cup (33\textsuperscript{th}, 34\textsuperscript{th} and 35\textsuperscript{th} led engineers of GSEA Design to design appendage in order to improve performance and stability of sailing catamarans of teams such as Oracle Team USA, Artemis Racing and Groupama Team France. Although, the balance of these sailing boats can be relatively stable when their hulls remain in the water, it will be more precarious when they fly. The hydrodynamic loading of an appendage can lead to large deformations. When this appears, a coupling between the hydrodynamic loading and the appendage shape is necessary to compute the equilibrium of an appendage. Consequently, a Fluid Structure Interaction (FSI) model should be applied. From a structural engineering point of view, the aims of GSEA Design engineers with this tool is to :

- Know the appendages equilibrium in water flow
- Get closer to the real load-cases applied on the boat
- Optimize appendage structure to stabilize boat fly

Since an optimization process requires several FSI calculations, a fast method in term of CPU time is needed. The flow around an appendage is generally three dimensional and lead mostly to bending and torsional deformations and displacements. A 3D RANS method for the flow computation coupled with a 3D finite element method could be appropriate to model the physical phenomenon. Nevertheless, these methods are out of the scope of the paper due to their long CPU time. A faster and consistent manner to deal with the FSI calculation in order to optimize a structural design is the use of a non linear 3D lifting line [2] coupled with the Timoshenko beam element method [5]. Duport et al. [2] show very accurate results for small angle of attack compared to the 3D RANS method. Moreover, it can be shown that the bending and the torsional stiffness of the structure has a high impact on the equilibrium. Consequently, a pseudo-analytical method has been developed to evaluate these properties. GSEA Design has

\begin{figure}[h]
\centering
\includegraphics[width=0.4\textwidth]{figure1.png}
\caption{Groupama Team France AC45 Test (©Eloi Stichelbault / Groupama Team France)}
\end{figure}

\begin{figure}[h]
\centering
\includegraphics[width=0.4\textwidth]{figure2.png}
\caption{Artemis Racing AC45 Turbo (©Eloi Stichelbault / Artemis Racing)}
\end{figure}
Rémy Balze, Nedeleg Bigi, Kostia Roncin, Jean-B. Leroux, Alain Nême, Vincent Keryvin, Antoine Connan, Hervé Devaux and Denis Gléhen

Figure 3: Different types of appendages: Straight-Shape, C-Shape, T-Shape and L-Shape

Figure 4: Appendage coordinate system

thus developed a tool for appendage design using FSI calculation and called Sofia\textsuperscript{1}.

In a first part, the FSI method and the pseudo-analytical method to evaluate the bending and torsional stiffness are introduced. Then the iterative numerical scheme to reach equilibrium is presented. In a second part, two illustrating examples are presented, one on a simple FSI calculation and a second on the optimization of a dagger-board structural design in term of mass and stiffness.

2 APPENDAGE MODELING

2.1 Appendage Description

An appendage is a part of the boat beneath the hull. Classical appendages are rudders which enables to steer the boat and dagger-boards which counters the leeward thrust of sails (Figure 3).

Since the aim is to design a flying boat, appendages able to produce a vertical lift force are used: T-Shaped Rudder and L-shaped dagger-boards (Fig. 3). In addition to their vertical lifting force, they produce transverse components forces needed for the global equilibrium of the sailing boat.

An appendage can be seen as a simple supported beam. The appendage orientation is parametrized with three angles depicted in Fig. 4. Altitude over the free surface is also an important parameter that can modify the load-case distribution.

For a L-shaped foil, its vertical part of a foil is commonly called shaft whereas its horizontal part is called tip.

\textsuperscript{1}Structural Optimisation using Fluid-structure Interactions for Appendage design
2.2 Hydrodynamic Model

The fluid flow around the sections along the span, except near the tip, behaves essentially as two-dimensional flow for lifting bodies with high aspect ratio. However, the pressure difference between intrados and extrados creates a more and more significant flow component along the span as we approach the end of the tip. This creates a flow enrollment around the tip edge creating the so-called tip vortex, which can be source of significant loss in lift and energy. Consequently, the fluid flow around the appendage should be considered as three-dimensional to be accurate. This phenomenon is particularly well represented on a straight appendage with the so-called lifting line method proposed by Prandtl [1] in 1918.

However, Prandtl lifting line method is strictly applicable only on straight span appendage. Since appendages used on high performance sailing boats are curved in order to allow the boat to fly over water, the non linear 3D lifting line method proposed in [2] (see also [4]) is here applied. Indeed, this extension of the lifting line theory is an iterative numerical method which takes into account the evolution of the dihedral and sweep angles. In Figure 5, the fundamental difference between the Prandtl’s lifting line method [1] and the method proposed by Duport et al. [2] is that the bounded vortices are no longer aligned and parallel. The induced velocity at any collocation point may be dependent of the other bounded vortices. Then, using the iterative method presented in Anderson [3] to calculate the vorticity, the non-linearity of section lift coefficients, for example computed with 2D RANS method, can be taken into account.

2.3 Structural Model

The hydrodynamic model is coupled with the Timoshenko beam model [5], which takes into account shear forces.

Figure 6 shows the element description: an element joins the nodes I and J. At each node corresponds a section, a laminate and mechanical properties such as:
- Young and shear modulus
- Section and reduced section considering local coordinate system
- Bending and torsion inerties considering local coordinate system
Section angle of attack is highly coupled with bending and torsion of the appendage. As a result, an accurate evaluation of the bending and torsional stiffness of finite beam element model has a high impact on the load-case, and so on the FSI calculations. Consequently, a pseudo-analytical method has been developed in order to obtain a good evaluation of bending and torsional stiffness of a composite material section [5][6][7]. For instance, torsion stiffness of fairing is the same as thin profile [7] and can be expressed as:

\[ GI_{zz} = 4 \cdot S^* \sum_i \left( \frac{L_i}{G_i \cdot t_i} \right) \]  

The mechanical properties of section obtained with this pseudo-analytical method has been validated using 3D finite element analysis.

3 FLUID STRUCTURE INTERACTION NUMERICAL SCHEME

3.1 Classical Iterative Process

GSEA Design focused on hydro-elastic phenomena and FSI on lifting structures in water in order to provide answers to performance, efficiency and stability issues. Here, the fluid-structure interactions model developed is a quasi static and iterative process (Figure 7). At each iteration, an equilibrium is reached. The assumption are:
- Slender structure
- Sections are not deformable
- Small perturbations (linear stiffness matrix)

A hydrodynamic loading is firstly calculated, considering the appendage attitude in the fluid flow. This hydrodynamic loading is then applied to the non-deformed or deformed geometry of the beam model. The iterative process ends when the convergence criteria, between two successive fluid-structures loops, is lower than a user defined convergence criteria. At the end, the internal forces and deformed geometry calculated allow to design the appendage regarding stresses or stiffness.

3.2 Target Loads

A second type of calculation considers side force denoted by \( F_y \) and lift force denoted by \( F_z \) as inputs. *Sofia* is able then to optimize appendage position in the fluid in order to reach the target loads. It can give for instance the corresponding Rake and Yaw angles for the boat to
take off the free surface:

\[
\begin{pmatrix}
F_y(yaw, rake) \\
F_z(yaw, rake)
\end{pmatrix}
- \begin{pmatrix}
F_{y\text{-target}} \\
F_{z\text{-target}}
\end{pmatrix} = \begin{pmatrix}
0 \\
0
\end{pmatrix}
\]  \hspace{1cm} (2)

Equation (2) is solved numerically. This method works with all the developed hydrodynamic loading and calculation methods.

4 ILLUSTRATING EXAMPLE

In order to prepare the 35th America’s Cup, Franck Cammas and Groupama Sailing Team have decided to build a C Class catamaran (Figure 8). These catamarans are powered by a 27.8 m² wing-sail. They are 7.62 meters length and 4.20 meters width. Groupama C Class has been designed with a close collaboration between several architects and has won two times the Little Cup (2013 and 2015).
4.1 Fluid Structure Interaction Results

Each load-cases are characterized by a boat speed, an altitude over free surface and a Cant angle. *Sofia* can then calculate yaw and rake angles. The corresponding load-case enables to design this appendage (Fig. 9) regarding stiffness.

Figures 10 and 11 show respectively the bending moment and the shear stress along the span for two method of calculation. The first method in red line represents the results without FSI, *i.e.* only the first loop of the FSI iterative algorithm Fig. 7 is performed. The second method, in blue line, represents the result with full FSI calculation.

For this illustrating example, the FSI calculation leads to lower internal forces at the bottom bearing and at the elbow, the connection between the tip and the shaft. Therefore, by considering this more accurate loading distribution, the appendage design can be lighter.

In term of CPU time, the FSI computation takes around one minute on classical PC.

4.2 Optimization results

Optimization can be proceeded with *Sofia*. In this illustrating example, UD quantity in appendage stock is optimized, the objective is to minimize mass and to maximize stiffness regarding failure stresses. The multi-optimization results are plotted in Fig. 12 in term of Pareto-efficient frontier. The best designs are distributed near the red curve. It can be shown that a lighter appendage has a lower stiffness and a stiffer appendage is heavier. The final design is chosen in agreement with the requirement specifications and the designer experience.

The necessary calculation the Pareto-efficient frontier in Fig. 12 takes around 6 hours on classical PC.

5 CONCLUSION

A fast and efficient fluid structure interaction method has been presented. This method is based on an iterative algorithm using a 3D non-linear lifting line for the hydrodynamic loading and a modified beam element method. The beam element method has been modified in order to represent with good accuracy the bending and torsional stiffness. It has been shown that it is necessary to tune the initial position of the appendage in order to reach a target load. The
Figure 10: Bending moment distribution along the span

Figure 11: Shear force distribution along the span

Figure 12: Pareto-efficient frontier to optimize UD quantity along an appendage’s span
two illustrating examples shows that the method is fast in term of computation time, which is necessary to study a wide range of design.

A simple multi-objective results has been presented in order to optimize the appendage mass and stiffness. Further investigations to optimize the boat speed and stability can be performed with a velocity prediction program coupled to the presented FSI method.

ACKNOWLEDGEMENT

The authors would like to thank Artemis Racing and Groupama Team France for their help to validate this tool.

REFERENCES


APPLICATION OF A 6DOF ALGORITHM FOR THE INVESTIGATION OF IMPULSE WAVES GENERATED DUE TO SUB-AERIAL LANDSLIDES

ARUN KAMATH*, HANS BIHS AND ØIVIND A. ARNTSEN

* DEPARTMENT OF CIVIL AND ENVIRONMENTAL ENGINEERING
NORWEGIAN UNIVERSITY OF SCIENCE AND TECHNOLOGY
NTNU TRONDHEIM, NORWAY
e-mail: arun.kamath@ntnu.no

Key words: Landslide, Tsunami, 6DOF, CFD, REEF3D

Abstract. Inland water bodies such as lakes, rivers and streams are generally considered safe from extreme wave events. Such inland water bodies are susceptible to extreme wave events due to impact of aerial landslides, where a large mass of land impacts the water at high velocities, resulting in a sudden transfer of momentum to the water body. Similar events can occur due to an underwater landslide as well. The evaluation of such extreme events in inland water bodies and the impact of such extreme waves on the regions adjacent to the water body is essential to assess the safety of the constructions on the banks of the water bodies. The generation of extreme waves due to aerial and sub-aerial landslides depends on several parameters such as the height of fall, the composition of the impacting land mass and the bottom slope of the water body.

In this paper, the 6DOF algorithm implemented in the open source Computational Fluid Dynamics (CFD) model REEF3D is used to simulate the motion of a sliding wedge impacting the water free surface. This is used to represent a sliding landmass impacting water after a landslide event. The wedge is represented using a primitive triangular surface mesh and a ray-tracing algorithm is used to determine the position of the object with respect to the underlying grid. Further, the level set method is then used to represent the solid boundary. The motion of the wedge is obtained by propagating the level set equation. The interaction of the wedge with the free water surface is obtained in a sharp and accurate manner using the level set method for both the water free surface and the solid boundary. REEF3D uses a staggered Cartesian numerical grid with a fifth-order WENO scheme for convection discretisation and a third-order Runge-Kutta scheme for time advancement. With the higher-order methods and the level set method, the model can be used to calculate detailed flow information such as the pressure changes in the water on impact and the associated deformation of the water free surface. The accurate representation of these characteristics is essential for correctly evaluating the height and period of the generated extreme wave and associated properties such as the wave celerity and wave run up on the banks during the extreme event.
1 INTRODUCTION

The impact of large landmasses on confined interior water bodies can have disastrous consequences. The generation of tsunamis in the ocean due to natural activities such as earthquakes and volcanic eruptions have been studied from different perspectives, geological and hydrodynamic. On the other hand, the generation of an extreme tsunami-like wave in an inland water body due to landslides presents several interesting problems to investigate, model and obtain more insights. Landslide impact on water results in a sudden transfer of momentum from the falling landmass to water. This displaces a large amount of water while providing it sufficient energy that results in a large tsunami-like wave as the displaced mass of water approaches the shore. The fall event can also result in large run ups at the near shore that is upstream of the fall event. The waves generated by the landslide depends on several factors such as the physical properties of the sliding mass such as its density and volume, in addition to the local topographical features such as the water depth and bottom slope. The height of fall and the composition of the sliding mass also influence the wave generation.

Due to the large number of variables that can influence the tsunami-like wave generation due to a landslide event, numerical modelling of such events can be employed to simulate the different circumstances. With advances in computational power and high performance computing, a numerical model based on the Navier-Stokes equations can be used to calculate the impact of the moving land mass and the influence of this impact on water in a detailed manner.

In this study, the open-source CFD model REEF3D [1], developed at the Department of Civil and Environmental Engineering, NTNU Trondheim is used to simulate the impact of a slide with water. The model has been previously applied to study several marine engineering problems such as breaking wave kinematics [2], calculation of wave forces [3] [4] and floating bodies in waves [5]. In this paper, the 6DOF algorithm in REEF3D is used to model the motion of a wedge driven down a slide to simulate a sub-aerial landslide. The impact of the wedge with water and the calculated resulting free surface features are presented. The numerical results are compared to experimental data for the free surface elevations at various locations in the tank around the region of impact of the wedge.

2 NUMERICAL MODEL

The incompressible Reynolds-averaged Navier-Stokes (RANS) equations are the governing equations of the numerical model.

\[ \frac{\partial u_i}{\partial x_i} = 0 \] (1)

\[ \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + g_i \] (2)

where \( u \) is the velocity averaged over time \( t \), \( \rho \) is the fluid density, \( p \) is the pressure, \( \nu \) is the kinematic viscosity and \( g \) the gravity term. The convective terms of the RANS equations are discretized with the fifth-order WENO scheme by Jiang and Shu [6] in the conservative finite-difference framework. The Jacobi-Hamilton version of the WENO scheme [7] is used for the variables of the free surface algorithm. Time integration is carried out using a 2nd-order
The numerical model is based on a Staggered Cartesian grid making the implementation of higher order schemes easier and providing good pressure-velocity coupling. The CFL number is used to govern the time steps in the simulation with an adaptive time stepping method. This maintains the required time step for the stability of the simulation while being economical on the computational cost of the simulation.

Chorin’s projection method [9] method is used for pressure treatment and the resulting Poisson pressure equation is solved using the geometric multigrid PFMG preconditioned [10] BiCGStab solver [11] available through the high performance solver library HYPRE [12]. The level set method [13] is used to obtain the free surface and the level set function is reinitialised every time step using the PDE based method [14] to maintain its signed distance property.

The 6DOF algorithm is used to simulate the motion of the wedge. The geometry of the wedge is defined by a primitive triangular surface mesh resembling the STL format used as a standard in design and meshing software. The ray tracing algorithm [15] is used to determine the intersection of the underlying Cartesian grid with the surface mesh. Then, signed distance property of the level set function in the vicinity of the body is obtained with the standard reinitialization algorithm [16]. The ray-tracing algorithm provides a sharp representation of the solid-fluid interface with exact calculation of the distances close the solid boundary. The moving solid-fluid interface is treated with the ghost cell immersed boundary method [17]. Further details about the 6DOF algorithm and its implementation in REEF3D can be found in Bihs et al. [5].

3 RESULTS AND DISCUSSION

3.1 Disc entry

The 6DOF algorithm in REEF3D is validated for the popular disc entry problem, where a solid disc enters water under controlled fall. This case has been presented by several authors such as [18] and [19] as benchmark tests for their models. The non-dimensional parameters for the geometry, the initial boundary conditions and flow parameters for this case are as follows. The disc impacts the water surface travelling under a fixed vertical velocity of \( V = -1 \). The center of the disc with radius \( R = 1 \) is located at \( H = 1.25 \) over the free surface at \( t' = V t / H = 0 \).

The two-dimensional simulation domain is \( 30R \times 22R \) long and high respectively and carried out with a uniform mesh of \( dx = 0.025 \). The simulation is carried out with \( 1200 \times 880 \) grid cells. The acceleration due to gravity is \( g = 1 \), the density of the water \( \rho_{\text{water}} = 1 \), the density of the air \( \rho_{\text{air}} = 0.001 \), the viscosity of the water \( \nu_{\text{water}} = 0.001 \) and the viscosity of the air \( \nu_{\text{air}} = 0.018 \). The time step size is controlled with adaptive time stepping using a CFL number of 0.1.

The free surface location and contour for the velocity magnitude are shown in Fig. 1. The disc enters the water mass in Fig. 1a. Breaking waves are generated in a symmetric fashion, when the disc impacts the free surface. In general terms, the representation of the formation of these breaking waves needs a sufficiently fine mesh [20] and specifically in the current case [19]. In Fig. 1b, the post-breaking waves are seen moving away towards the side walls as the disc is further immersed into water. At \( t' = 3.0 \), the parted free surface reconnects over the upper surface of the disc as it is completely immersed in water in Fig. (1c). At \( t' = 4.0 \), the water mass that reconnected over the disc creates a vertical water jet after the impact of the disc. The free
surface location compares well with other results reported in literature.

Figure 1: Controlled fall of disc into water showing velocity magnitude contours

(a) $t' = 1.0$

(b) $t' = 2.0$

(c) $t' = 3.0$

(d) $t' = 4.0$
3.2 Impact of a sliding wedge on water

The impact of a sliding wedge in to a body of water is simulated in this section. The experimental results presented by [21] are used to compare the numerical results for the free surface elevation. The wedge has a width of 0.61 m and a height of 0.46 m and a length of 0.91 m. The wedge is placed such that the top of the wedge is 0.454 m above the still water level. The wedge is then allowed to slide into water in a controlled manner. The numerical setup is presented in Fig. (2). Wave gages are placed at various location in the numerical flume to measure the free surface elevation due to the tsunami-like wave caused due to the impact of the wedge. A grid size of $dx = 0.05$ m is used in the simulation.

The calculated free surface elevations at the different wave gages placed in the numerical flume are compared with the experimental data in Fig. (3). The numerical results show a reasonable agreement with the experimental observations. At WG 1, the first wave is slightly underestimated while the following wave is overestimated in Fig. (3a). This observed difference is reduced at WG 2 placed a little in front of WG 1 as seen in Fig. (3b). The numerically calculated free surface elevation at wave gages WG 3 and WG 4, placed along the line $y = 2.46$ m are seen to show a good agreement with the experimental data in Figs. (3c) and (3d). The free surface elevations calculated at wave gages WG 5, WG 6, WG 7 and WG 8 in Figs. (3e), (3f), (3g) and (3h) respectively underestimate the free surface elevation in the first 1 s of the simulation as the wedge impacts the water surface but the agreement between the numerical and experimental results improves afterwards. The wave gages WG 9 and WG 10 also underestimate the first wave while the second wave is represented correctly as seen in Figs. (3i) and (3j).
(a) WG 1 at $x = 2.73$ m, $y = 1.85$ m

(b) WG 2 at $x = 3.64$ m, $y = 1.85$ m

(c) WG 3 at $x = 2.73$ m, $y = 2.46$ m

(d) WG 4 at $x = 3.64$ m, $y = 2.46$ m

(e) WG 5 at $x = 1.39$ m, $y = 2.94$ m

(f) WG 6 at $x = 1.77$ m, $y = 2.94$ m

(g) WG 7 at $x = 2.15$ m, $y = 2.94$ m

(h) WG 8 at $x = 1.39$ m, $y = 2.46$ m
Figure 3: Comparison of the numerical results to the experimental data for the free surface elevation in the wave tank due to the sliding mass impact with water

(i) WG 9 at $x = 1.77$ m, $y = 2.46$ m

(j) WG 10 at $x = 2.15$ m, $y = 2.46$ m

(a) $t = 0.40$ s

(b) $t = 0.60$ s

(c) $t = 0.75$ s

(d) $t = 0.88$ s
Figure 4: Calculated evolution of the free surface in the numerical flume due to impact of the wedge showing velocity magnitude contours
The evolution of the free surface due to the impact of the wedge is presented in Fig. (4). The impact of the wedge with the free water surface is just initiated at \( t = 0.40 \) s in Fig. (4a). Half the front surface of the wedge is immersed in water after \( t = 0.60 \) s in Fig. (4b). The top of the wedge is seen to be at the same level as the initial free surface at \( t = 0.75 \) s in Fig. (4c). The wedge is just completely immersed in water at \( t = 0.88 \) s as seen in Fig. (4d). The generation of the water jet due to the immersion of the wedge into the water body is seen in Figs. (4e), (4f), (4g) and (4h). The propagation of the extreme wave after the wedge has reached the bottom of the slope is seen in Figs. (4i) and (4j).

4 CONCLUSIONS
- The numerical model REEF3D is used to simulate the controlled motion of a wedge in to water and the numerical results show a reasonable agreement with experimental data.
- Future studies on refined grids and for submerged slides will be carried out.

REFERENCES


DEVELOPMENT OF A FLUID STRUCTURE COUPLING FOR COMPOSITE TIDAL TURBINES AND MARINE PROPELLERS

POL MULLER* AND FABIAN PÉCOT*

* DCNS Research / SIREHNA
Technocampus Océan
Rue de l'Halbrane, 44340 Bouguenais, France
e-mail: pol.muller@sirehna.com, fabian.pecot@sirehna.com

Key words: Fluid Structure Coupling, Propeller, Tidal Turbine, Composite

Abstract. The recent strong development of composite blades for propellers and tidal turbines has been driven both by the reduction of mass compared to metallic materials and by the impact of the deformations of blades on the increase of their performances (as shown for instance in [1]). The gain in performances concerns the efficiency of the propeller or turbine, the mitigation of the risk of cavitation and, by extension, the reduction of noise and vibrations. In order to help the designer to make the appropriate choices during the early stages of the design, new numerical tools like a fluid-structure coupling for "heavy" fluid are necessary in addition to existing numerical and experimental methods. This paper focuses on the development of such a fluid-structure coupling algorithm for tidal turbines and marine propellers. The main objectives for the blades are: to get an accurate estimation of the deformations and stresses under a hydrodynamic load, to include these predictions in a design process in order to increase efficiency and reliability and finally to optimize the hydrodynamic shape and inner structure. In terms of software development the main target is to provide an efficient tool which can be integrated in an optimisation environment for preliminary to intermediate design phases, therefore with low resource consumption, fast execution time, easy file setup and fully scripted for an automated execution in command line.

1 FLOW SOLVER

The flow solver used by the present coupling algorithm is PROCAL, a 3D Boundary Element Method (BEM) based on potential flow theory, developed by the CRS organisation [2]. The PROCAL code computes the wetted and cavitating potential flow around blade geometries operating in an input wake field. It solves the steady or unsteady potential flow problem for an arbitrary number of rotating and non-rotating surfaces. The lifting bodies generate a wake geometry which can be described by an input mesh or by wake generation routines which have been implemented for rotating bodies such as propeller or tidal turbine blades. In order to account for viscous effects which are not considered in a potential flow theory, a post-processing computation of the forces is made using empirical corrections based on flat plate skin friction coefficients.

Assuming the flow to be incompressible and irrotational, the flow velocity \( V \) can be defined as a function of a potential \( \Phi \) as in (1). This potential is split into two contributions (2) where
\( \varphi_\infty \) is the potential of the undisturbed flow, and \( \varphi \) is the disturbance potential to be solved, which satisfies the Laplace equation (3). The boundaries of the domain consist of the blade and hub surfaces \( S_B \) and \( S_H \) on which the kinematic boundary condition (4) is applied.

\[
\vec{V} = \nabla \Phi \tag{1}
\]

\[
\Phi = \varphi_\infty + \varphi \tag{2}
\]

\[
\nabla^2 \varphi = 0 \tag{3}
\]

\[
\frac{\partial \varphi}{\partial n} = -\vec{V}_\infty \cdot \vec{n} \tag{4}
\]

At infinity, the disturbance due to the body on the flow vanishes to zero. In addition, the lifting surfaces are generating a wake surface \( S_W \) which consists of vortex sheets being shed from the trailing edges. The normal velocity is null on this surface and the pressure is continuous across it.

Assuming that the potential is null within the bodies, i.e. within the surfaces \( S_B \) and \( S_H \), the second Green’s theorem applied to the computational domain shows that the resolution of the flow can be reduced to the unknowns at the boundaries only (5).

\[
\varphi_p = \frac{1}{2\pi} \int_{S_B \cup S_H} \left[ \varphi_q \frac{\partial}{\partial n_q} \left( \frac{1}{R_{p,q}} \right) - \frac{\partial \varphi_q}{\partial n_q} \frac{1}{R_{p,q}} \right] dS \tag{5}
\]

Without going into the mathematical resolution of this equation and its implementation in PROCAL, which can be found in [3], we can tell that the strengths of the singularities for the sources and potentials are assumed to be constant on each panel. A mesh is built with quadrilateral panels on the blade and hub surfaces (Figure 1).

**Figure 1**: Example of a panel mesh on a propeller including only one blade wake surface for better visibility.

This flow solver has been extensively validated within CRS working groups for a large range of propellers with various shapes, number of blades and diameters, for model scale and full scale. Although it is suitable for computing the flow on blades either in uniform inflow or in a
non-uniform wakefield, it is used in the present coupling algorithm only in axisymmetric inflow.

The coordinate system used by PROCAL and recommended by the International Towing Tank Conference (ITTC) is chosen as the general coordinate system for the present coupling algorithm. The first axis $X$ is longitudinal (along the rotation axis) towards upstream, the third axis $Z$ is coincident with the blade reference line toward the blade tip and the second axis $Y$ completes the right handed reference frame, see Figure 2.

![Figure 2: Propeller reference system, in [4].](image)

2 STRUCTURE SOLVER

The commercial solver ABAQUS 6.13 [5] distributed by "Dassault Systèmes" is used in the present coupling algorithm.

2.1 Type of elements

Various elements can be applied to describe the structure: volume elements, shells or thick shells. Considering that the propeller blade is made of composite material distributed in thin layers (plies) with orthotropic properties, it is not easily described by volume elements which should have the same thickness as each ply or group of plies with similar properties. In order to grant as much freedom as possible to the designer, the algorithm should not assume that the plies can be subdivided into groups with similar properties, but should consider each ply separately. The thickness of such plies can easily be below 0.5 mm which implies that the total number of elements will be incompatible with a low computer resource consumption and with low computation time. On the other hand, the volume elements are an adequate representation of the actual geometry, both in terms of inner structure and in terms of outer shape. This outer layer is the boundary between hydrodynamics and structure, where the information is exchanged between the flow solver and the structure solver (see chapter 3). Therefore, it is of main interest for the algorithm to preserve the geometry of this boundary.

The shell elements provide an excellent efficiency in terms of computer resources because the number of elements is much less than when using volume elements, and because the mathematical modelling of the material is simplified. In this case, the blade is assumed to be
infinitely thin and is replaced by its mid plane. The formulation of shell sections in ABAQUS can handle the distribution of plies in each element by describing the ply material, orientation and thickness. Unfortunately the geometrical link between the shell and the outer boundaries of the structure is not represented. This means that the algorithm has to transfer the hydrodynamic pressure applied on the suction and pressure sides to the shell elements and in return has to compute the deformation of the outer surfaces of the blade. The transfer of pressure can be relatively easily handled with projection but the deformation of the outer surface based on the shell deformation can introduce uncertainties, especially in areas with large thickness or curvature variations, such as leading edges and trailing edges.

A reasonable intermediate between volume and shell elements is the thick shell element in ABAQUS which combines a low number of elements for a complete description of the blade, with simplified mathematical description of the material, and an element which has actual nodes on the outer surface of the blade. The transfer of information such as pressure and deformations on the faces is therefore possible without any specific pre/post processing and the computation time remains low enough to be used in an optimisation environment. One remaining issue is that the material has to be applied as a single stack on the element. It is well supported by ABAQUS as presented on Figure 3, but the user has to specify the exact stack of plies. As the thickness of the blade is not constant, the number of plies varies from one element to another so the user almost has to define a new stack for each element. This can be easily tackled with an automated pre-processing based on the actual element thickness, directly computed from the node coordinates, which selects the appropriate plies to be applied to the element. This type of element has been selected for the modelling of the blade structure within the coupling algorithm.

Figure 3: Example of ply stack representation in ABAQUS, in [5].

2.2 Definition of the material

The composite material is supposed to be orthotropic and distributed in plies. The target material is a Fibre-Reinforced Plastic, such as CFRP or GFRP (carbon or glass fibres). In ABAQUS this type of material can be defined as a stack of plies with variable mechanical properties, thicknesses and orientations of the fibres. It is therefore possible to define a stack for a given thick shell element. The orthotropic mechanical properties of the material are given as inputs together with the thickness of each ply. The main issue concerns the orientation of the material within the ply. A detailed computation of an actual composite blade would need to use
a wrapping tool which can be included in some CAD suites. The typical purpose of such a tool is both to provide a local coordinate system (LCS) of a ply for any point in the structure and to check that the skew and twist of the fibres will be acceptable in areas with large curvatures. As the present algorithm is dedicated to optimisation tasks in a fully automated environment, the usage of such a wrapper is not realistic. One has to define the LCS according to a chosen convention. A literature review does not emphasize a clear unique convention. For instance in [6] the fibre directions are defined in the local spanwise coordinate system, whereas in [7] the LCS is the same as the general coordinate system. The main consequence is that, for a given fibre orientation, the resulting model with these two different conventions will not be identical and may produce different results in terms of mechanical behaviour (stresses, deformations and hydrodynamic forces). For the present coupling algorithm another convention has been used, taking into account the strength and weaknesses of the above described conventions. It is not likely that a fibre with a zero degree orientation will follow the local spanwise direction, especially for high skew blades. This would also result in high skew of the fibre which is not favourable for the strength of the material. On the other hand it is likely that the fibre will be twisted or skewed in order to follow the general shape of the blade. A unique LCS is therefore not chosen. The convention used in the present algorithm is such that the third axis of the LCS is the normal to the shell element towards upstream, the first axis is in the XZ plane of the general coordinate system and the second axis completes the right handed reference frame, see Figure 2. It is a balance between a global coordinate system and a system purely based on a blade element geometry.

\[ \text{Figure 4: Local coordinate system for the definition of the fibre orientation in a ply.} \]

3 COUPLING ALGORITHM

The general algorithm is based on successive iterations on the displacement of the mesh nodes and of the pressure on the mesh panels, until convergence, as presented on Figure 5. It is solved in static condition, i.e. in an axisymmetric inflow, without any influence of a dynamic deformation of the blade or of added mass.
3.1 Mesh

As already mentioned the pressure and displacement are exchanged over a boundary surface between fluid and structure which is the envelope of the blade, namely its pressure and suction sides. It is therefore important to preserve as much as possible the information exchanged on this surface in order to limit any numerical discrepancy that would be introduced by interpolations or extrapolations. The simplest solution to cope with this potential issue is to use a coincident mesh both for the BEM method and for the structure solver. The hydrodynamic pressure is computed by the flow solver at the panel centres: this information can be directly transferred to the facets of the structure elements. In return the deformations of the structure elements are computed at the elements nodes: this can be directly used to update the mesh read by the flow solver. In consequence the meshes used by the flow and structure solvers have at least their surface nodes in common from the first to the final iteration. There is no need to build a new mesh after each iteration which saves computation time, prevents interpolations and preserves the history of stresses in structure elements. One can also easily check that, at each iteration, on the one hand the pressure computed by the flow solver is the same as the one used as input for the structure solver, and on the other hand that the node coordinates of the structure and fluid meshes are the same. In order to keep the history of stresses in structure elements from one iteration to the following, the structure computation for an iteration is restarted from the previous solution simply by updating the input pressure field.
3.2 Code implementation

The implementation of the coupling algorithm has been made using python scripts. These scripts can handle each step of the computation process in an automated way. The steps covered are: meshing of the blade for the flow solver starting from its geometric description, building the structure mesh, applying the material on the structure elements, building the control scripts for the flow solver and for the structure solver, running the solvers, reading and converting the pressures and displacement from one solver to the other, updating the fluid mesh and finally check the convergence. The convergence of the algorithm is based both on hydrodynamic forces (thrust and torque on the blades) and on structure displacements. All relevant parameters of the simulation have been gathered in a single general input file which has to be filled by the user, and are then allocated to the corresponding input files of the solvers. As a matter of fact isotropic materials are considered as a special case of orthotropic materials therefore the coupling algorithm can handle metallic materials for instance.

In terms of computing performances on a desktop computer, one iteration takes between 1 minute for a low skew blade (up to a skew angle of 15 degrees with 1 500 to 2 000 panels on the blade surface) up to 5 minutes for a high-skewed blade (with skew angle higher than 45 degrees and 10 000 panels on the blade surface). The computation time is almost equally divided between the flow resolution and the structure deformation. The targets in terms of software development are therefore achieved: the tool can be fully scripted for an automated execution in command line, with low resource consumption, fast execution time and easy file setup.

4 TEST CASE

The test case which has been selected is described in [7] and [8]. It is based on the high-skew propeller DTNSRDC 4498 [9] with 5 blades and a diameter of 1 foot (Figure 6). Starting from the original geometry, [7] and [8] define a material (CFRP) with two different orientation sequences. Based on deformation computations, they define a new "pre-deformed" geometry with one orientation sequence, which should have the original DTNSRDC 4498 shape under a given hydrodynamic load. This sums up to a total of three different blades which are summarized in the Table 1. For a complete description of the orientation sequence convention, please refer to [8]. One should emphasize that the orientation sequence of the 4498_1 propeller produces deformations which are close to an equivalent isotropic material, whereas the second orientation sequence gives a clear dominant direction of deformations.

Reference results of propellers 4498_1, 4498_2 and 4498_3 are presented in [8]. They consist of both computation results and experimental measurements of thrust coefficient, torque coefficient and blade deformation for various inflow velocities and rotation speeds, for a total of 54 different cases.
Figure 6: Overview of the 4498 propeller.

Table 1: Summary of test cases

<table>
<thead>
<tr>
<th>Case id</th>
<th>Orientation sequence</th>
<th>Pre-deformed</th>
</tr>
</thead>
<tbody>
<tr>
<td>4498_1</td>
<td>$[-45^\circ/90^\circ/45^\circ/0^\circ/-45^\circ/90^\circ/45^\circ/0^\circ/-45^\circ/90^\circ/45^\circ/0^\circ]$</td>
<td>No</td>
</tr>
<tr>
<td>4498_2</td>
<td>$[45^\circ/90^\circ/45^\circ/45^\circ/45^\circ/0^\circ/0^\circ/0^\circ/0^\circ/0^\circ/45^\circ/2]$</td>
<td>No</td>
</tr>
<tr>
<td>4498_3</td>
<td>$[45^\circ/90^\circ/45^\circ/45^\circ/45^\circ/0^\circ/0^\circ/0^\circ/0^\circ/0^\circ/45^\circ/2]$</td>
<td>Yes</td>
</tr>
</tbody>
</table>

5 RESULTS

The following results have been obtained after a proper mesh independence study, for the flow solver, the structure solver and the coupling of both. It appears that the necessary mesh refinement is higher for the convergence of the structure deformations than for the convergence of the hydrodynamic forces on this specific case.

5.1 Blade deformations

In order to compensate for any offset that could introduce unexpected inaccuracy in the experimental setup, the deformations are compared for each propeller versus the deformations of the 4498_1. The description of the experimental setup for measuring the blade deformations is not detailed on the possible gap of the thrust bearing which may introduce offsets into the blade deformation measurement along the axial direction. These deformations are measured in this direction only by means of video cameras which are used to track the displacement of markers on the leading and trailing edges of the blades, at 95% of the blade radius. Results are presented in Figure 9. For the leading edge (two first rows of the figure) the order of magnitude of the difference with experimental measurements is less than 0.2 mm and the sorting of the blades by order of deformation intensity is preserved, which is of major importance for an optimisation study. On the other hand the results are less satisfactory on the trailing edge (two last rows of the figure): both the order of magnitude of the displacement and the sorting are significantly different from the leading edge results. It is important to notice that the trailing edge thickness (up to 80% of the chord length) is less than 1 mm which means that the number of plies in this area is 2 or 4. There is a significant manufacturing uncertainty in this area. In addition the theoretical profile used in the computation has a zero thickness at the trailing edge which is of course not feasible in reality. These manufacturing uncertainties on the trailing edge thickness may produce significant differences in terms of deformations.
Figure 7: 4498_3 blade deformations at J=0.600 (20Hz) seen from the suction side (left) and from the leading edge (right), coloured by deformation amplitude and magnified x10.

5.2 Hydrodynamic forces

For the 4498_1 propeller, with an orientation sequence such that the material is almost isotropic, the hydrodynamic forces induced by the deformations at the design point are within 1 to 2% on thrust coefficient compared to experimental results, and 1 to 10% for torque coefficient, depending on the rotation rate.

For the 4498_2 propeller, the thrust and torque coefficients induced by the deformations at the design point are within 2% for the highest rotation rate (13Hz) and 10 to 15% for the lowest rotation rate (7Hz). For the latter case one can notice that the Reynolds number at 0.7R is well below the recommended values by the ITTC. The flow on the blade should be mostly laminar which is not accurately resolved. It is also possible that the actual material orientation is inadequately represented by the chosen LCS convention.

For the 4498_3 propeller, the results are in the same order of magnitude as for the 4498_2 propeller.

Examples of open water results (best and worse) are presented on Figure 10.

Figure 8: Pressure field on the 4498_3 blade at J=0.600 (20Hz).
6 CONCLUSIONS AND PERPECTIVES

The main objectives of the coupling algorithm are split into computing performances, user friendliness, and result accuracy. The computing performances are reasonable enough to be integrated in an optimisation environment where a large number of blade variants should be computed, both in terms of geometry modifications and material stacking sequences. The user friendliness is also very decent as there is only one main control text file with less than a dozen of parameters to fill in. This can also be filled or modified in an automated way within an optimisation environment. The results accuracy need to be improved and extended to other tests cases. The identified sources of discrepancies are: uncertainties on the actual geometry of the manufactured propeller model, small diameter (1 foot) which implies small deformations, high-skewed blade and laminar flow which are difficult to compute with BEM software, usage of thick shell structure elements which are simplified compared to volume elements, possible discrepancies in the fibre orientation definition. Although this coupling algorithm has some perfectible features, it is still possible to use it in a preliminary or intermediate design phase, with optimisation of the blade geometry and fibre orientations.

This coupling algorithm is dedicated to propeller and tidal turbine blades in an axisymmetric inflow. An extension to a non-uniform inflow, such as a ship wake field of a current inflow profile close to the sea bed would extend the possible use of this tool.

REFERENCES

Figure 9: Overview of the blade deformations at 95% of the radius.
Figure 10: Examples of open water results.
MODELLING OF HYDRODYNAMIC LOADS ON AQUACULTURE NET CAGES BY A MODIFIED MORISON MODEL

PÅL T. BORE∗, JØRGEN AMDAHL† AND DAVID KRISTIANSEN‡

∗ Centre for Autonomous Marine Operations and Systems (NTNU AMOS)
Exposed Aquaculture Operations (EXPOSED)
Department of Marine Technology
Norwegian University of Science and Technology, NTNU
NO-7491 Trondheim, Norway
e-mail: paal.takle.bore@ntnu.no

† Centre for Autonomous Marine Operations and Systems (NTNU AMOS)
Exposed Aquaculture Operations (EXPOSED)
Department of Marine Technology
Norwegian University of Science and Technology, NTNU
NO-7491 Trondheim, Norway
e-mail: jorgen.amdahl@ntnu.no

‡ Exposed Aquaculture Operations (EXPOSED)
SINTEF Ocean
NO-7465 Trondheim, Norway
e-mail: david.kristiansen@sintef.no

Key words: Hydrodynamic load models, Aquaculture structures, Net cages

Abstract. A modified Morison model is presented for calculation of hydrodynamic forces on aquaculture net cages. The model is based on a simple method for conversion of "screen model force coefficients" to approximate equivalent directional dependent Morison coefficients. The motivation for this is that experimentally obtained force coefficients for net panels are generally presented as screen model force coefficients, while commercial analysis software are often restricted to a Morison model. Based on the screen model force coefficients defined by Løland’s formulas, the method is implemented in the nonlinear finite element program USFOS. Analyses with the modified Morison model are performed, investigating the viscous forces on a net panel for different inflow angles. The numerical results are benchmarked against Løland’s original screen model, showing good agreement for all inflow angles. Comparison with the classical Morison model is made to illustrate the advantage of the proposed method. The model is also applied for calculation of hydrodynamic forces on the net of a real aquaculture structure exposed to a steady current. Again, the results are compared to Løland’s original screen model, showing an almost exact reproduction of the global drag force.
1 INTRODUCTION

Proper hydrodynamic modelling is extremely important in order to accurately estimate the response of a structure exposed to the marine environment. For aquaculture structures, this is a challenging task. Typical fish farms represent a highly elastic system where fluid-structure interaction effects are important. Calculation of the forces by state-of-the-art computational fluid dynamics (CFD) methods are prohibitively expensive since the number of twines for a fish farm net is in the order of ten millions, calling for rational methods for assessment of the hydrodynamic loads on aquaculture net cages [1].

Generally, two different types of hydrodynamic models are applied for calculation of viscous forces on nets or screens: (1) Morison type and (2) screen models. The advantage of approach (1) lies in its simplicity and widespread use for analysis of slender marine structures – literally all relevant analysis tools include the option to select such a hydrodynamic model. On the downside, it largely over-predicts the drag force for large inflow angles on a net panel as it is not able to capture important fluid-structure interaction effects, which typically are dependent on the inflow angle. In addition, a drag model based on the cross-flow principle cannot be justified for inflow angles larger than about 45 degrees [1]. Application of Morison models for calculation of hydrodynamic forces on aquaculture net cages are found in e.g. Tsukrov et al. [2], Fredriksson et al. [3], Moe et al. [4, 5], Fredheim [6] and Zhao et al. [7]. Bi et al. [8] used it in combination with a porous media fluid model to also simulate the effect the presence of the net cage has on the flow.

In approach (2), the net is divided into several net panels/screens, and the hydrodynamic force is decomposed into a drag and a lift component. By defining the unit normal vector of the net panel, it is possible to take into account the angle between the incoming flow and the net panel, resulting in more accurate force estimation. Another advantage is that experimentally obtained force coefficients for net panels are typically presented as “screen model force coefficients” [9, 10, 11, 12]. The main drawback is that screen models are often not available in commercial analysis software. Presentation and application of screen models for calculation of hydrodynamic forces on net cages are found in e.g. Løland [9], Kristiansen and Faltinsen [1, 13], Huang et al. [14] and Lader and Fredheim [15].

In this paper, a modified Morison model will be presented for calculation of hydrodynamic loads on aquaculture net cages. The model is based on a simple method for converting screen model force coefficients to approximate equivalent directional dependent Morison coefficients. The motivation for this is that experimentally obtained force coefficients for net panels are generally presented as screen model force coefficients, while most analysis tools are restricted to a Morison type hydrodynamic force model. The method allows for direct application of experimentally obtained screen model force coefficients in a Morison model, including the force dependence on the inflow angle relative to the net panel.

2 HYDRODYNAMIC MODEL

In the following, the proposed hydrodynamic model will be described. For convenience, we will first outline the basic principles behind both the classical Morison type models and the screen models.
2.1 The Morison model

For calculation of hydrodynamic loads on slender marine structures, Morison’s equation [16] is frequently used. It gives us the cross-flow force on a member based on the cross-flow principle [1]. In the case of a fixed structure, the total hydrodynamic force is split into an inertia term, representing the Froude-Krylov force and the diffraction force, and a drag term, representing the viscous drag forces. It is assumed that the water particle velocity and acceleration in the region of the structure do not differ significantly from the value at the cylinder axis; an assumption generally valid for $D/\lambda < 0.2$, where $D$ is the structural diameter and $\lambda$ is the wave length [17]. For a vertical rigid circular cylinder, the classical Morison’s equation tells us that the total force normal to the cylinder axis, i.e. the horizontal force, $dF$ on a strip of length $dz$ can be written as [18]:

$$dF = \rho \pi D^2 \frac{dz}{4} C_m \cdot a_1 + 2 \rho C_d Ddz \cdot u | u |$$

where $\rho$ is the water density, $D$ is the diameter of the cylinder, $C_m$ and $C_d$ are the inertia and drag coefficients, respectively, and $a_1$ and $u$ are, respectively, the horizontal water particle acceleration and velocity.

If the structure moves, the acceleration of the structure must be accounted for in the added mass part of the inertia term, and the velocity of the structure in the drag term. The general expression for the total force per unit length normal to the axis of the considered structural member can then be written as [17]:

$$F_n = \rho C_m dV \cdot a_n - \rho (C_m - 1) dV \cdot \ddot{\eta}_n + \frac{1}{2} \rho C_d dA \cdot (u_n - \dot{\eta}_n) | u_n - \dot{\eta}_n |$$

here, $u_n$ and $a_n$ are, respectively, the wave particle velocity and acceleration perpendicular to the member, $\ddot{\eta}_n$ and $\dot{\eta}_n$ are the time derivatives of the member motion perpendicular to the member and $dA$ and $dV$ are the exposed area and displaced water per unit length.

Even though it is not considered in the original Morison’s equation, the presence of a mean lift force can be handled by introducing

$$\bar{F}_l = \frac{1}{2} \rho C_l dA \cdot u_n^2$$

where, $\bar{F}_l$ is the mean lift force, which is orthogonal to $u_n$ in the cross-flow plane, and $C_l$ is the lift coefficient. $\bar{F}_l$ is zero for a single body in infinite fluid when the body is symmetric about the axis parallel to the direction of $u_n$ [18]. The mean lift force is therefore seldom considered for slender marine structures as they typically have circular cross-sections. The motion of the structure can then be handled by replacing $u_n$ in Eqn. (3) with the relative normal velocity.

An illustration of the different force contributions acting on a structural member is shown in Fig. 1, where $f_N$ corresponds to the force predicted by Morison’s equation. The tangential force $f_T$, which is primarily due to shear forces (skin friction), is generally negligibly small for net threads [1].

A simple example, illustrating the use of a Morison type force model for calculation of the cross-flow forces on a net exposed to a steady current, is shown in Fig. 2. The mean lift force on
Figure 1: Definition of normal force \( f_N \), tangential force \( f_T \) and lift force \( f_L \) on an inclined slender structural member exposed to a water particle velocity \( V \). Illustration from [19].

Element level is not considered. Note that even though only the drag term of Morison’s equation contribute to the two element forces \( F_1 \) and \( F_2 \), the drag force on the inclined cylinder (\( F_1 \)) is actually contributing to a lift force on the net seen from a global perspective.

Figure 2: (a) Illustration of a Morison type force model applied to two twines in a steady current of magnitude \( U_\infty \). (b) Illustration of a net with twine diameter \( d_w \) and twine length \( l_w \). Illustrations from [1].

2.2 The screen model

The basic assumption in screen models is that the net cage can be divided into several net panels, or screens, and the model aims to provide a good estimate of the total force acting on each of these net panels. The screen models are primarily made for analysis of net cages in current, but due to a quasi-static assumption (\( KC \gg 1 \)), they are also applicable in waves [1].

The mean drag and lift force on a net panel/screen are typically dependent on force coefficients which magnitude depends on (among other things) the inflow direction relative to the screen. They can be written as:

\[
F_{d,\text{screen}} = \frac{1}{2} \rho C_{d,\text{screen}}(\theta) A' \cdot U_{\text{rel}}^2
\]  

(4)
\[ F_{l,\text{screen}} = \frac{1}{2} \rho C_{l,\text{screen}}(\theta) A' \cdot U_{rel}^2 \]  \hspace{1cm} (5)

where,
- \( F_{d,\text{screen}} \) force on the net panel in direction of the local (relative) inflow.
- \( C_{d,\text{screen}} \) drag coefficient of the screen.
- \( F_{l,\text{screen}} \) force on the net panel perpendicular to the local (relative) inflow.
- \( C_{l,\text{screen}} \) lift coefficient of the screen.
- \( A' \) area of the net panel.
- \( U_{rel} \) relative inflow velocity.
- \( \rho \) density of water.
- \( \theta \) angle between the relative inflow direction and the net normal vector in the direction of the flow.

The panels are characterized by their solidity ratio \( S_n \) and their orientation relative to the (relative) inflow, denoted by the angle \( \theta \) \cite{1}. The solidity ratio is the ratio of the area projected by the threads of a screen to the total area of the net panel. The definition of the angle \( \theta \) is illustrated in Fig. 3, together with the unit normal vector \( \hat{n} \) and the relative inflow unit vector \( \hat{u} \). The direction of the drag on a panel is defined in the direction of the relative inflow unit vector \( \hat{u} \). The lift direction is perpendicular to \( \hat{u} \), and is defined by the cut between the plane defined by \( \hat{n} \) and \( \hat{u} \), and the normal plane of \( \hat{u} \). Mathematically, the lift unit vector \( \hat{l} \) can be expressed as:

\[ \hat{l} = \frac{\hat{u} \times (\hat{n} \times \hat{u})}{|\hat{u} \times (\hat{n} \times \hat{u})|} \]  \hspace{1cm} (6)

where \( \hat{n} \) is the unit normal vector of the panel defined such that it always point into the same half-space as the relative flow velocity, i.e.:

\[ \hat{n} = \text{sign} \ (n \cdot \hat{u}) \ n \]  \hspace{1cm} (7)

The reason why the total force on a net panel is not in the inflow direction, is due to deflection of the flow through the screen \cite{1}.

The drag and lift coefficients, \( C_{d,\text{screen}} \) and \( C_{l,\text{screen}} \), are primarily determined based on experiments with net panels in steady flow. The most important parameters, governing the magnitude of these coefficients, are found to be the solidity ratio \( S_n \) and the inflow angle \( \theta \) \cite{9}. Løland \cite{9} and Aarsnes et al. \cite{20} have developed analytical formulas for the drag and lift coefficients of screens as functions of the mentioned parameters. The model presented by Kristansen and Faltinsen \cite{1}, also takes into account the Reynolds number.

The screen models consider the viscous drag and lift forces on the net panels. Inertia forces can be included in a similar manner as in the Morison models, see e.g. \cite{14}. Even though viscous forces dominate, experiments with net panels exposed to waves have concluded that inertial forces on net structures are significant \cite{21}. However, few results on the magnitude of the inertia coefficients exist, and typical values for circular cylinders are thus generally applied.
2.3 Proposed hydrodynamic model

2.3.1 Note on the structural model

The net consists of millions of individual threads and twines [1]. Direct modelling thus involves a huge number of elements. This is not feasible for a structural analysis due to the enormous computational time and resources such a model would have required. A simplified model of the net with a coarser mesh must therefore be applied to keep the computational time within reason. The elements of this coarse mesh are modelled as circular cylinders, which are assigned an equivalent cross-sectional area, adding up the areas of the individual threads each element represents, so that the cross-sectional area is conserved. It is important to realize that such a procedure does not conserve the exposed area found in the drag term of Morison’s equation (see Eqn. (2)), meaning that the modelled net will have a smaller solidity ratio than the physical net. In the examples of the proposed hydrodynamic model, the elements have been modelled by three-dimensional beam elements. The reason for this choice is that the applied analysis software is optimized for such elements. The method is however equally applicable for truss and spring elements, which are more commonly used for netting structures.

2.3.2 The modified Morison model

The screen models gives us global directional dependent drag and lift coefficients for a whole net panel in accordance with Eqn. (4) and (5). If these are to be applied in a Morison model, the screen model drag and lift coefficients must be converted into equivalent Morison coefficients on element level, giving the same total force on the net panel. This is not a straight forward procedure as Morison’s equation uses the relative water particle velocity normal to the members in addition the element area, while the screen model uses the relative velocity and the net panel area. The lift and drag directions are also somewhat differently defined, and it is not common that the Morison coefficients are functions of the inflow angle, as is the case for the screen.
model coefficients. A simple method to convert screen model force coefficients to be used in a modified Morison model, applying directional dependent coefficients, has thus been made. The idea behind the method can be explained by a simple example.

Consider the physical, plane net in Fig. 2b exposed to a steady current of magnitude $U_{\infty}$. Denote the area of this net panel as $A'$. In an assumed structural model, this net panel is simply modelled by two elements; one vertical with a projected area $A_V$, representing the vertical threads, and one horizontal with projected area $A_H$, representing the horizontal threads, i.e. similar as in Fig. 2a (the length of the horizontal member is in the direction normal to the paper plane in this figure). In a screen model, the drag forces on the net is now determined by the physical properties of the net, the flow characteristics and the screen model drag coefficient $C_{d,\text{screen}}$, in accordance with Eqn. (4). We now want to obtain equivalent drag coefficients to be used in a Morison model, giving the same total force on the net. If the current is normal to the net panel, the only difference between the Morison and the screen model is the area which the force coefficients are normalized by. The equivalent Morison drag coefficient $C_d$, is then simply obtained as:

$$C_d = \frac{C_{d,\text{screen}}}{S_{n_{\text{model}}}}, \quad \text{where } S_{n_{\text{model}}} = \frac{A_H + A_V}{A'} \quad (8)$$

Now, the net panel is given an angle $\theta$ relative to the current, as illustrated in Fig. 2a. The drag on the vertical member is then reduced by a factor of $\cos^2 \theta$. Defining the element normal vector in the same direction as the normal vector of the net panel, it is noted that the current now arrives at an cross-sectional angle $\phi = \theta$ on the horizontal member relative to the element normal vector (see Fig. 4a for an illustration of $\phi$). To compensate for the force reduction on the vertical member, we want to modify the drag coefficient of the horizontal member in such a way that the total obtained force is compatible with the screen model. This means that we have to solve the following equation with respect to the drag coefficient of the horizontal member, $C_{d,H}$:

$$\frac{1}{2} \rho C_{d,\text{screen}}(\phi) A' U_{\infty}^2 = \frac{1}{2} \rho C_{d,H}(\phi) A_H U_{\infty}^2 + \frac{1}{2} \rho C_{d,V} A_V U_{\infty}^2 \cos^2 \theta$$

Noting that $A' = \frac{A_H + A_V}{S_{n_{\text{model}}}}$ and introducing the crude approximation that the drag coefficient of the vertical element $C_{d,V} = \frac{C_{d,\text{screen}}(\phi)}{S_{n_{\text{model}}}}$, as in Eqn. (8), the following result is obtained:

$$C_{d,H}(\phi) = \frac{C_{d,\text{screen}}(\phi)}{S_{n_{\text{model}}}} \left(1 + \frac{A_V}{A_H} \sin^2 \phi \right) \quad (9)$$

The latter term in Eqn. (9) can be thought of as a directional-dependent correction factor.

In the three-dimensional case, the inflow can also arrive at an cross-sectional angle on the vertical member relative to the element normal vector. This is accounted for in a similar way as in Eqn. (9), and the drag coefficient of the vertical member $C_{d,V}$, can be expressed as:

$$C_{d,V}(\phi) = \frac{C_{d,\text{screen}}(\phi)}{S_{n_{\text{model}}}} \left(1 + \frac{A_H}{A_V} \sin^2 \phi \right) \quad (10)$$

This approach is exact for inflow normal to the net panel and parallel to the net panel. For intermediate inflow angles, it is only an approximation. The model will slightly overpredict...
Figure 4: (a) Definition of the cross-sectional angle $\phi$ in the element local coordinate system. The local Z-axis is defined in the direction of the element normal vector $\mathbf{n}$ (the net panel plane is thus in the local XY-plane). (b) Visualization of a circular cylinder with a direction dependent drag coefficient as applied in section 3.1.

the drag forces normal to the net panel, and underpredict the forces parallel to the net panel compared to a screen model.

We now introduce lift coefficients on element level, following Eqn. (3). The magnitude of these coefficients are determined based on the screen model lift coefficients, and are both for the vertical and horizontal member, simply expressed as:

$$C_l(\phi) = \frac{C_{L,\text{screen}}(\phi)}{S_{\text{model}}}$$  \hspace{1cm} (11)

As the drag term in Morison’s equation already has contributed to a lift force (see Fig. 2a), the directional-dependent correction factor is not included in Eqn. (11). This is seen to improve the overall performance of the model, counteracting the over- and underprediction mentioned above. Note that the magnitude of the lift coefficients of a net panel are much less than the drag coefficients.

The procedure for implementing the modified Morison model on a whole net cage is given below.

1. Define an element unit normal vector $\mathbf{n}$ in the same direction as the local net panel normal vector for all net elements. One of the principal axes of each element’s local coordinate system must then be defined in the direction of its element normal vector to ensure a consistent definition of the element cross-sectional angle $\phi$ (see Fig. 4a).

2. Estimate directional dependent screen model force coefficients for the physical net based on empirical formulas such as Løland’s formulas [9], model tests or other methods giving coefficients on the format of Eqn. (4) and (5). The physical properties of the net should be used here (not the modelled). As the net will serve as a surface for biofouling [22], a larger solidity than the clean net solidity should be applied in design calculations.
3. Calculate the local solidity ratio $S_n_{model}$, and the local area ratios $\frac{A_H}{A_V}$ and $\frac{A_V}{A_H}$ for the modelled net. Depending on the modelling strategy, one might get different values of $S_n_{model}$ and the area ratios for different parts of the net. Uniform meshing is recommended to simplify this step.

4. Convert the screen model force coefficients to approximate equivalent directional dependent Morison coefficients on element level as:

$$C_{d,V}(\phi) = \frac{C_{d,screen}(\phi)}{S_n_{model}} \left(1 + \frac{A_H}{A_V} \sin^2 \phi \right)$$  \hspace{1cm} (12)

$$C_{d,H}(\phi) = \frac{C_{d,screen}(\phi)}{S_n_{model}} \left(1 + \frac{A_V}{A_H} \sin^2 \phi \right)$$  \hspace{1cm} (13)

$$C_l(\phi) = \frac{C_{l,screen}(\phi)}{S_n_{model}}$$  \hspace{1cm} (14)

where,

- $C_{d,V}$ element drag coefficient of vertical net elements.
- $C_{d,H}$ element drag coefficient of horizontal net elements.
- $C_l$ element lift coefficient of both vertical and horizontal net elements.
- $A_V$ local modelled exposed area of the vertical net elements.
- $A_H$ local modelled exposed area of the horizontal net elements.
- $\phi$ element cross-sectional angle (see Fig. 4a).

5. The inertia forces are included as in a regular Morison model. In lack of experimental data, standard values for the inertia coefficient of circular cylinders could be applied, e.g. $C_m = 2$. The effect of biofouling should be accounted for in design calculations.

6. When the net cage is exposed to a current, the shielding effect from the net panels upstream, will lead to a reduced incident current velocity on the net panels and other structural components downstream. This should be accounted for. Løland [9] has suggested the following formula for this current velocity reduction factor $r$, which should be assigned to the downstream members:

$$r = \frac{u}{U_\infty} = 1.0 - 0.46C_{d,screen}$$  \hspace{1cm} (15)

The derived drag, lift and inertia coefficients are now input to a modified Morison model, allowing both directional dependent coefficients and inclusion of the lift coefficient. The applied Morison model must be able to account for the motion of the structure. To properly account for hydroelasticity, the element local coordinate system must "follow" the displacements and rotations of each element. An illustration of an element with a directional dependent drag coefficient is shown in Fig. 4b.
2.3.3 Comments

As previously mentioned, the proposed conversion method is exact for inflow normal to the net panel and parallel to the net panel. For intermediate inflow angles, it is only approximate. The main approximation is related to the fact that the obtained force coefficients are directional dependent on the considered elements cross-sectional angle, only. Denoting the cross-sectional inflow angle on a vertical member as $\phi_V$ and the inflow cross-sectional angle on a neighbouring horizontal element as $\phi_H$, exact conversion of the screen model coefficients would require the element force coefficient to be functions of both $\phi_V$ and $\phi_H$. Such an approach would however require that the force coefficients are re-calculated at every time step, making it a complex and less attractive solution.

It is recommended to model the net by a rectangular mesh with an aspect ratio close to one (i.e. the ratio $\frac{A_V}{A_H}$ should not be too high or too low). The main reason for this is that very high aspect ratios of the mesh is somewhat problematic for the conversion of the lift coefficients. In addition, real nets typically have close to square meshes. A trapezoidal mesh could be applied for the bottom net as long as it does not deviate too much from a square shape.

The proposed modified Morison model requires the local modelled solidity ratio $S_{n_{\text{model}}}$ and the local area ratio $\frac{A_V}{A_H}$ (and its inverse) for conversion of the screen model force coefficients. The implementation of the model becomes a lot easier if the local values of $S_{n_{\text{model}}}$ and $\frac{A_V}{A_H}$ are varied as little as possible for the different part of the net, i.e. application of uniform meshing. For the side net, uniform meshing is easily performed. For the bottom net, this could be more challenging, but one should keep in mind that the forces on the bottom net are generally small compared to those on the side net. As long as the mesh size and aspect ratio of the modelled bottom net are not varied too much, average values could be used. Representation of the vertical net element on the side net with lengths exponentially increasing with depth (shorter lengths near the sea surface to better resolve the wave kinematics), will complicate the implementation. Numerical investigations by Kristiansen and Faltinsen [13] have, however, concluded that the difference in the obtained forces on net cages in waves by using uniform vertical meshing is negligibly small compared to using exponential vertical meshing.

A last comment is related to the implementation of the directional dependent lift coefficient, $C_l(\phi)$. Depending on how the element lift direction is defined in the applied analysis software, one must assure that the lift force is actually calculated in the correct direction. Typically, one has to change the sign of $C_l(\phi)$ for every 90° of $\phi$.

3 RESULTS – VALIDATION OF THE MODEL

The presented modified Morison model has been implemented in the nonlinear finite element program USFOS [23]. The drag and lift coefficients are based on screen model force coefficients found by application of Lølands formulas [9], i.e.:

\[
C_{d,\text{screen}}(\theta) = 0.04 + (-0.04 + 0.33S_n + 6.54S_n^2 - 4.88S_n^3) \cos \theta
\]

\[
C_{l,\text{screen}}(\theta) = (-0.05S_n + 2.3S_n^2 - 1.76S_n^3) \sin 2\theta
\]

The above formulas are valid for nets with a solidity ratio in the range of 0.13 – 0.31.
Two examples will be given to demonstrate the performance of the model. In the first one, the viscous forces on a fixed net panel for different inflow angles will be investigated. In the second, the method will be applied for calculation of the viscous forces on a real aquaculture structure exposed to a steady current.

3.1 Viscous forces on a rigid net panel

A net panel of dimensions 1.0 m $\times$ 1.0 m was modelled in USFOS. The net is characterized by a solidity ratio $S_n = 0.15$, and is attached to rigid frame as shown in Fig. 5a. This is similar to the set-up typically applied for experimental determination of screen model force coefficients. The frame was modelled so that it only provided stiffness, but did not attract hydrodynamic loads.

The aim is to investigate the viscous forces on the net panel by application of the modified Morison model for different inflow angles. The drag and lift coefficients in the modified Morison model are based on Løland’s formulas given by Eqn. (16) and (17), and converted according to Eqn. (12), (13) and (14). The drag coefficient of a net element is shown in Fig. 4b.

The properties of the net panel are given in Tab. 1. The solidity of the modelled net is taken equal at the solidity of the ”real” net, i.e. $S_{n,\text{model}} = S_n$. A square, uniform mesh with $N_H \times N_V = 10 \times 10$, where $N_H$ is the number of horizontal threads and $N_V$ is the number of vertical threads, is applied.

In the numerical simulations, the net panel is exposed to an inflow velocity $U$ of magnitude 1 m/s. The direction of $U$ is given by the angles $\beta$ and $\alpha$, which are defined in Fig. 5b together with the global coordinate system. The net panel plane is in the global $YZ$-plane and the net panel normal vector $n$ is defined in the negative $X$-direction. The angle $\beta$ is the inflow angle in the $XY$-plane, and corresponds to the heading of a wave or a current relative to the net panel normal vector $n$. The angle $\alpha$ is equivalent to the phase angle of the water particles in a long, regular, deep water wave with a heading defined by $\beta$. Inertia forces are not considered, and the analyses were run as quasi-static.

<table>
<thead>
<tr>
<th>Net panel properties</th>
<th>Inflow properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>$S_n$ 1.0 m$^2$ D$_{\text{threads}}$ Mesh $\rho$</td>
<td>$</td>
</tr>
<tr>
<td>0.15 15 mm square 10×10 1024 kg/m$^3$ 1.0 m/s</td>
<td></td>
</tr>
</tbody>
</table>

Analyses with four different inflows, each with a constant $\beta$ angle, were run. These corresponds to $\beta$ angles of 0°, 30°, 60° and 90°. The forces have been recorded over one wave period, i.e. all 360° of $\alpha$, for each $\beta$ value. $\beta = \alpha = 0^\circ$ corresponds to inflow in the positive X-direction, normal to the net panel. The results obtained for the modified Morison model were benchmarked against Løland’s original screen model. In addition, the results were compared to the classical Morison model. In the original screen model, the angle between the net panel normal vector and the incoming velocity vector, $\theta$, is defined as $\theta = \cos^{-1}(\cos \alpha \cos \beta)$. In the classic Morison model, $C_d = 1.15$ has been applied, following [4]. The forces have been decomposed into their X-, Y- and Z-component, and are presented in Fig. 6, 7 and 8, respectively. To better illustrate
Figure 5: (a) Net panel with dimensions 1x1 metres as modelled in USFOS. (b) Definition of the global coordinate system, the net normal vector $\mathbf{n}$ and the incoming water particle velocity $\mathbf{U}$ with inflow angles $\beta$ and $\alpha$. $\beta$ is the inflow angle in the XY-plane, and corresponds to the heading of a wave/current relative to $\mathbf{n}$, while $\alpha$ corresponds to the phase angle of a long, regular deep water wave with heading defined by $\beta$.

Each force components relative importance, the force range on the vertical axis of the plots is kept constant for each $\beta$ value.

Figure 6: Comparison of the forces in X-direction. The angles $\alpha$ and $\beta$ are defined in Fig. 5b.

As seen from the plots in Fig. 6, 7 and 8, the performance of the proposed Morison model is good, and it is able to accurately recreate the Løland screen model forces. Generally, the model slightly overpredicts the forces normal to the net panel (x-direction), with a correspondingly small underprediction of the forces parallel to the net panel (y- and z-direction). The classical
Morison model results are primarily included to illustrate the differences between the models. For this model, the overprediction of forces for inflow parallel to the net panel (i.e. for $\beta = 90^\circ$ and all Z-forces with $\alpha = \pm 90^\circ$) are clearly observed. Underprediction of the forces normal to the net panel compared to the screen model are also seen in Fig. 6.

3.2 Viscous forces on a rigid aquaculture structure in current

A rigid, semi-submersible aquaculture structure was modelled in USFOS, applying the modified Morison model as the hydrodynamic model for the net. Specifically, the considered structure is SalMar/Ocean Farming’s concept “Ocean Farm 1”, illustrated in Fig. 9.
The side net is modelled by a uniform, approximately square mesh. The bottom net is modelled by a trapezoidal mesh which is not uniform. Related to calculation of the hydrodynamic coefficients, the bottom net has been split in two zones; one inner and one outer, separated at half the radius of the cage. Average values of $S_{n_{\text{model}}}$, $A_{H}$ and $A_{V}$ are used within each zone.

The hydrodynamic coefficients of the net elements are calculated based on Løland’s formulas, Eqn. (16) and (17), and converted by Eqn. (12), (13) and (14). The shielding effect from the upstream net panels, causing a reduced incident current velocity on all downstream elements, is taken into account using Eqn. (15).

A simple validation of the global performance of the modified Morison model is made by exposing the structure to a steady current of magnitude $U = 0.75$ m/s. The total in-line hydrodynamic force acting on the netting structure is then computed. This is compared with calculations of the total in-line hydrodynamic force, using Løland’s screen model directly. The results are found in Tab. 2.

Table 2: The total in-line hydrodynamic force acting on the netting structure of the Ocean Farming concept when exposed to a current of magnitude 0.75 m/s.

<table>
<thead>
<tr>
<th></th>
<th>Modified Morison model</th>
<th>Løland</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>742 [kN]</td>
<td>748 [kN]</td>
</tr>
</tbody>
</table>

The agreement between the total force predicted by the modified Morison model and that by Løland’s screen model is good. This suggests that the global performance of the modified Morison model is satisfactory.

4 CONCLUSIONS

A modified Morison model has been proposed for calculation of hydrodynamic forces on netting structures. The model is based on an introduction of directional dependent drag and lift coefficients. The magnitude of the coefficients, is determined by screen model force coefficients. By a simple method, these are then converted to equivalent Morison coefficients, giving approx-
imately the same total force on a net panel as a screen model. This allows for direct application of screen model force coefficients in software restricted to a Morison type hydrodynamic model.

Based on the screen model force coefficients defined by Løland’s formulas, the model was implemented in the nonlinear finite element program USFOS. Analyses with the modified Morison model were performed, investigating the viscous forces on a net panel for different inflow angles. The results were benchmarked against Løland’s original screen model, showing good agreement for all inflow angles. Comparison with the classical Morison model was also made, illustrating the advantage of the proposed method. A validation of the global performance of the model on a real aquaculture structure was also made, comparing the obtained in-line hydrodynamic forces on the netting structure with Løland’s screen model. The agreement was good.

Further validation of the model should be performed, particularly on flexible net cages.

5 ACKNOWLEDGEMENT

This work has been carried out at the Centre for Autonomous Marine Operations and Systems (NTNU AMOS) and the centre for Exposed Aquaculture Operations (EXPOSED). The Norwegian Research Council is acknowledged as the main sponsor of both NTNU AMOS and EXPOSED. This work was supported by the Research Council of Norway through the Centres of Excellence funding scheme, project number 223254 – NTNU AMOS, and through the the Centres for Research-based Innovation funding scheme, project number 237790 – EXPOSED.

REFERENCES


MONOLITHIC COUPLING OF RIGID BODY MOTION AND THE PRESSURE FIELD IN FOAM-EXTEND

Inno Gatin∗, Vuko Vukčević, Hrvoje Jasak

University of Zagreb, Faculty of Mechanical Engineering and Naval Architecture, Ivanu Lučića 5, Zagreb, Croatia
e-mail: inno.gatin@fsb.hr, vuko.vukcevic@fsb.hr, hrvoje.jasak@fsb.hr

Key words: CFD, foam–extend, Rigid Body Motion, Seakeeping

Abstract. In this paper a monolithic algorithm for coupling rigid body motion equations and pressure field within a Finite Volume framework is presented. Monolithic coupling enables fewer number of pressure–velocity loops per time–step, thus reducing the required computational time. The presented method is compared to conventional partitioned coupling approach in terms of computational efficiency and accuracy. The results are compared to experimental data as well.

1 INTRODUCTION

The interaction between a floating rigid body and free surface waves is a problem often encountered in the field of naval hydrodynamics. The complexity of the problem encouraged utilisation of viscous Computational Fluid Dynamics (CFD) codes, where the interaction between the two–phase, viscous flow and the rigid body can be fully resolved. The forces acting on the rigid body are a result of combined pressure and viscous effects which the fluid exerts upon the boundary of the rigid body. The oscillatory nature of forces exerted on the body due to surface waves are dominated by the change in pressure forces, while viscous effects play a minor role in the oscillatory motion. Hence, given the segregated nature of the pressure–velocity coupling in modern CFD algorithms, it is necessary to properly resolve the interaction between the body motion and the pressure field in each time–step.

Conventional approach for resolving rigid body motion and fluid flow coupling solves the body motion equations once per pressure–velocity coupling loop. In transient simulations, multiple pressure–velocity coupling loops are necessary in order to obtain convergence of fluid flow. The introduction of body motion increases the number of necessary pressure–velocity coupling loops per time–step, since it represents a flow–dependent moving boundary. It is the intention to avoid the increase in the number of pressure–velocity coupling loops.

In this work the pressure–body motion coupling is resolved on the linear solver level of the pressure equation. Hence, monolithic coupling between the pressure equation and rigid body motion equations is achieved. The monolithic coupling leads to a fully resolved pressure–body motion coupling at the end of every pressure equation solution. Hence, no additional pressure–velocity loops are needed on the account of the rigid body motion.
The performance of the novel approach is compared to the conventional partitioned approach on a seakeeping benchmark case of a KCS hull from Tokyo 2015 Ship Hydrodynamics workshop [1]. The comparison shows that similar accuracy in terms of motion and added resistance is obtained while using a smaller number of pressure–velocity correctors per time–step. The results are compared to experimental values in order to validate the method.

Naval Hydro software pack based on foam-extend is used which is specialised for simulating large scale, two–phase flows in Finite Volume (FV) framework. Regular waves are imposed using SWENSE (Spectral Wave Explicit Navier Stokes Equations) method [2, 3], which allows accurate and robust two–phase simulations. Implicitly redistanced Level Set method is used for interface capturing, while Ghost Fluid Method [4] is employed to discretise the dynamic free surface boundary condition, enhancing the stability of the simulation.

The paper is organised as follows. First, the mathematical formulation of the monolithic pressure–body motion coupling is presented. Second, the comparison of the developed method against a conventional partitioned coupling scheme is shown, followed by a comparison with the experimental results. Finally, a conclusion is given.

2 MATHEMATICAL MODEL

Governing equations for two–phase, viscous, incompressible flow are given by the conservation of linear momentum and mass:

$$\frac{\partial \textbf{u}}{\partial t} + \nabla \cdot (\textbf{u} - \textbf{u}_M) \textbf{u} - \nabla \cdot (\nu_e \nabla \textbf{u}) = -\frac{1}{\rho} \nabla p_d,$$

$$\nabla \cdot \textbf{u} = 0,$$

where \(\textbf{u}\) presents the velocity field, \(\textbf{u}_M\) is the velocity of the relative grid motion [5], \(\nu_e\) is the effective kinematic viscosity, while \(\rho\) is the respective phase density. \(p_d\) is the dynamic pressure calculated as \(p_d = p - \rho g \cdot \textbf{x}\), where \(p\) denotes the total pressure, \(g\) is the gravity constant and \(\textbf{x}\) is the radii vector.

The rigid body motion can be taken into account in the continuity equation, Eq. (2), by accounting for the change of body boundary velocity \(\delta \textbf{u}_b\):

$$\nabla \cdot \textbf{u} + \nabla \cdot (\delta \textbf{u}_b) = 0.$$

By discretising Eq. (1) and Eq. (3) in the FV method fashion [6], Eq. (3) becomes:

$$\sum_f s_f \left( \frac{1}{a_P} \right)_f \left( \nabla p_d \right)_f = \sum_f s_f \left( \frac{\textbf{H}(\textbf{u}_N)}{a_P} \right)_f - \sum_f s_f \delta \textbf{u}_bf,$$

where \(f\) presents the cell face index, \(s_f\) is the face surface normal vector with a magnitude corresponding to the area of the face, \(a_P\) is the diagonal contribution of the discretised momentum equation, Eq.(1), while \(\textbf{H}(\textbf{u}_N)\) presents a linear function of the neighbouring cell velocities \(\textbf{u}_N\) which stems from the off–diagonal and source contribution of the discretised momentum equation. Eq. (4) is a conventional pressure equation, with an additional term on the right hand side which accounts for the change in the body boundary velocity. This way the pressure is coupled to the rigid body motion on the equation level.
In order to obtain the change of boundary velocity $\delta \mathbf{u}_b$, the rigid body motion equations need to be integrated:

$$
\frac{\partial \mathbf{v}}{\partial t} = \frac{\mathbf{F}}{m},
$$

$$
\frac{\partial \mathbf{\omega}}{\partial t} = \mathbf{I}^{-1} \cdot \left( \mathbf{M} - \mathbf{\omega} \times (\mathbf{I} \cdot \mathbf{\omega}) \right),
$$

where $\mathbf{v}$ denotes the translational velocity of the centre of mass, $\mathbf{F}$ is the external force exerted on the body and $m$ stands for the mass of the body. $\mathbf{\omega}$ is the angular velocity vector, $\mathbf{I}$ is the inertia tensor of the rigid body, while $\mathbf{M}$ denotes the external moment. The external force and moment are calculated by integrating the pressure and viscous stress on the body boundary:

$$
\mathbf{F} = \sum_{bf} s_{bf} p_{bf} + \mathbf{F}_v,
$$

$$
\mathbf{M} = \sum_{bf} r_{bf} \times s_{bf} p_{bf} + \mathbf{M}_v,
$$

where $bf$ stands for the body boundary face index, $s_{bf}$ is the normal surface vector of the boundary face with the magnitude corresponding to the surface area of the face, $p_{bf}$ is the total pressure acting on the boundary face. $r_{bf}$ is the radii vector of the face centre with respect to the centre of gravity. $\mathbf{F}_v$ and $\mathbf{M}_v$ denote the viscous portion of the force and moment acting on the rigid body boundary, respectively, which are treated explicitly.

Finally, the change of the boundary face velocity $\delta \mathbf{u}_{bf}$ is obtained as the difference of the translational and rotational velocity:

$$
\delta \mathbf{u}_{bf}^n = \mathbf{v}^n - \mathbf{v}^{n-1} + \left( \mathbf{\omega}^n - \mathbf{\omega}^{n-1} \right) \times \mathbf{x}_{bf},
$$

where $n$ denotes the linear solver iteration. Hence the equation set forming the pressure–rigid body motion coupling, which comprises Eq. (4) and Eq. (5) is closed by relations expressed with Eq. (6) and Eq. (7). These relations are evaluated at the level of pressure equation linear solver, Eq. (4), where the updated pressure solution is used to reintegrate the rigid body motion equations, Eq. (5).

3 NUMERICAL METHODS

In this work the Naval Hydro software pack is used, which is based on foam–extend CFD open–source software. Naval Hydro provides specialised numerical solvers for simulating large–scale, two–phase, viscous and turbulent flows. Ghost Fluid Method is used to discretise the dynamic and kinematic free surface boundary conditions [4], which eliminates the parasitic air currents present in many two–phase CFD codes. SWENSE method [2, 3] is employed to impose an arbitrary wave solution into the CFD model, where only the difference between the imposed flow solution and the full nonlinear solution is solved for. SWENSE enhances the accuracy of wave propagation in CFD and enables coarser temporal and spatial resolution while minimising the diffusion and dispersion of the wave field propagating through the CFD domain.

In this work a preconditioned Conjugate Gradient [7] method is used for solving the linear system arising from the discretised pressure equation. Fifth-order Cash-Karp embedded Runge–Kutta scheme with error control and adjustable time-step size [8] is used for rigid body motion integration.
In this section, a regular head wave seakeeping case is considered. The wave and ship velocity setup correspond to case C5 from the Tokyo Workshop on CFD in Ship Hydrodynamics [9], where the model scale KCS hull is used. In order to investigate the possible benefits of the new method, the two coupling strategies for pressure–rigid body motion coupling are compared, where the number of pressure–velocity loops per time–step is varied. Simulations using 2, 4 and 8 pressure–velocity loops are carried out, with four pressure correctors used per one pressure–velocity loop in all simulations.

The same computational mesh is used in all simulations, where half of the domain is simulated due to the symmetry of the phenomenon. Extremely coarse spatial discretisation is used, counting only 600,000 cells. Temporal resolution is set to 400 time steps per encounter wave period. The ship is free to pitch and heave, while a fixed surge velocity is prescribed. Nine items are compared in total: mean, first order amplitudes and first order phases of total resistance coefficient $C_T$, heave $z$ and pitch $\phi$.

The comparison is shown in Fig. 1, where all nine items are compared with respect to the number of pressure–velocity loops per time step. The first row of graphs show the comparison for the total resistance coefficient $C_T$. The second row shows dimensionless heave, while the third row shows dimensionless pitch. $\eta$ denotes the regular wave amplitude and $k$ stands for the wave number. The mean of an item is indicated with the subscript 0, and the first order amplitude with subscript 1. The first order phases are denoted with $\gamma$, where the subscript denotes the item the phase corresponds to. For most items, the monolithic approach exhibits low sensitivity with respect to the number of pressure–velocity loops. Moreover, with two pressure–velocity loops per time–step the monolithic approach produces similar results to the partitioned approach with eight pressure–velocity loops. This is true for all items related to resistance, first order amplitude of heave and pitch and for the first order phase of heave. Hence, the monolithic coupling enables accurate estimation of the most important seakeeping items, i.e. mean of total resistance and first order amplitudes of motion, with four times smaller number of pressure–velocity loops per time–step. For mean of pitch and heave, and for the first order phase of pitch, the two coupling strategies exhibit similar convergence patterns. It should be noted that mean of pitch $\phi_0$ has very small absolute values which cannot be captured with the coarse spatial discretisation that was used in the simulations.

Fig. 2 shows the required computational time per time–step for the monolithic and partitioned approach with respect to the number of pressure–velocity loops per time–step. The monolithic approach requires more computational time for the same number of pressure–velocity loops due to the additional integrations of the rigid body motion equation. However, since only two instead of eight pressure–velocity loops can be used with the monolithic approach, a total saving in computational time of a factor by 2.4 is achieved.

In order to validate the method, the results of both methods obtained with eight pressure–velocity loops are compared to the experiment. Table 1 shows the relative errors of all items for two coupling strategies. The relative error is computed as $E = (EFD - CFD)/EFD$, where CFD represents the result obtained using the computational method, while EFD stands for experimental result. The subscript denotes the item to which the relative error corresponds to. Given the extremely low spatial resolution used in these simulations, relative errors are satisfac-
Figure 1: Comparison of seakeeping results of partitioned and monolithic approach for pressure–rigid body motion coupling, with respect to the number of pressure–velocity loops per time–step $N_{nCorr}$.

The monolithic and partitioned approach have similar errors, which validates the present approach, since the partitioned approach has been thoroughly validated and verified in previous publications [10, 11].

5 CONCLUSION

Monolithic pressure–rigid body motion coupling method in FV numerical framework is presented in this paper. The new approach is intended for pressure dominated fluid–rigid body interaction problems, which are often encountered in the field of computational naval hydro-
Inno Gatin, Vuko Vukčević and Hrvoje Jasak

Figure 2: Comparison of required computational time per time–step with respect to the number of pressure–velocity loops per time–step $N_{nCorr}$.

Table 1: Result comparison with experimental data.

<table>
<thead>
<tr>
<th></th>
<th>Monolithic</th>
<th>Partitioned</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_{CT0}$, %</td>
<td>-4.95</td>
<td>-4.90</td>
</tr>
<tr>
<td>$E_{CT1}$, %</td>
<td>15.85</td>
<td>15.96</td>
</tr>
<tr>
<td>$E_{\gamma CT1}$, %</td>
<td>14.20</td>
<td>14.29</td>
</tr>
<tr>
<td>$E_{z0}$, %</td>
<td>-2.86</td>
<td>-3.60</td>
</tr>
<tr>
<td>$E_{\gamma z1}$, %</td>
<td>8.48</td>
<td>8.60</td>
</tr>
<tr>
<td>$E_{\phi 0}$, %</td>
<td>6.37</td>
<td>6.59</td>
</tr>
<tr>
<td>$E_{\gamma \phi 1}$, %</td>
<td>16.09</td>
<td>18.50</td>
</tr>
<tr>
<td>$E_{\phi 1}$, %</td>
<td>11.41</td>
<td>11.44</td>
</tr>
<tr>
<td>$E_{\gamma \phi 3}$, %</td>
<td>5.77</td>
<td>5.76</td>
</tr>
</tbody>
</table>

dynamics. The method is tested on a seakeeping case and compared in terms of accuracy and performance against the conventional partitioned approach. Results are compared to experimental data to validate the approach.

The comparison of the monolithic and partitioned approach shows that monolithic approach requires fewer pressure–velocity loops per time–step for the pressure–rigid body motion coupling to be properly resolved. Only two pressure–velocity loops are sufficient to obtain similar accuracy to the partitioned approach where eight pressure–velocity loops are employed. The reduction in the number of pressure–velocity loops per time–step reduces the computational time by a factor of 2.4. Comparison with experimental data validates the monolithic approach, where similar accuracy is achieved to the partitioned approach.

The monolithic pressure–rigid body motion coupling presents an advance coupling algorithm, which enables considerable acceleration in computational time. It is applicable to many problems in the field of naval hydrodynamics where the naval object can be considered as rigid.
REFERENCES


NUMERICAL INVESTIGATION OF STRONG ADDED MASS EFFECT FOR FLUID-STRUCTURE CALCULATIONS APPLIED TO MOVING HYDROFOILS

Emmanuel LEFRANÇOIS
Roberval Laboratory CNRS UMR 7337
Sorbonne Universités - Université de Technologie de Compiègne
CS 60319, 60203 Compiègne, France
e-mail: emmanuel.lefrancois@utc.fr

Key words: 2D Panel Method, fluid structure interaction, hydrofoil dynamics, added mass

Abstract. This paper presents a corrected partitioned scheme for investigating fluid-structure interaction (FSI) that may be encountered by lifting devices immersed in heavy fluid such as liquids. The purpose of this model is to counteract the penalizing impact of the added mass effect on the classical partitioned FSI coupling scheme. This work is based on an added mass corrected version of the classical strongly coupled partitioned scheme presented in [1]. Results show that this corrected version systematically allows convergence to the coupled solution with no dependency on fluid density. The fluid flow model considered here uses a non-stationary potential approach, commonly termed the Panel Method. The advantage of this kind of approach is twofold: first, in restricting itself to a boundary method and, second, in allowing an added mass matrix to be estimated as a post-processing phase.

1 CONTEXT AND INTEREST OF THIS STUDY

The research presented in this article focuses on the development of a numerical tool for investigating fluid-structure interactions (FSI) between a fluid flow that is not confined (infinite) and a current turbine with blades. Many similarities may be observed from its aerial version (wind-mill) but a major point of concern results from a fluid density 800 times higher than in the air. The constant search for an optimal solution (by increasing size and reducing mass) inevitably leads to flexible behavior resulting from hydrodynamic loads, and this flexible behavior may have serious impacts on the efficiency of the device.

Since simplifying is part of the process of understanding, the FSI model may here without loss of generality be restricted to a 2D airfoil placed in a flow and having two degrees of freedom (dof), namely plunging and pitching motions. This assumption is justified by the fact that the physical phenomena that occur around the cross-flow section of a blade are quite similar to those encountered by lifting airfoils in two dimensions (2D). The Panel Method approach [2, 3] is of particular interest for fluid flow calculations around lifting device due to the fact it has been originally designed for. This potential approach is restricted to incompressible and irrotational...
flows, however if completed by a Kutta condition, it can be extended to lifting flows. The FSI approach is here based on a partitioned coupling with a dedicated solver for each of the two physics (namely fluid flow and structure dynamics). Exchanges take place regularly between the two solvers via a coupling scheme \cite{4, 5} that is based on successive solutions produced by the fluid and structure solvers. The coupling is said to be loosely coupled partitioned if only one shot (that is to say a single computation) per time step is required for each field, and strongly coupled partitioned if an iterative procedure is used to ensure convergence of the coupled solution \cite{6}. In an industrial context, the biggest advantage that partitioned coupling has over monolithic coupling (with a single solver) is the modularity of the approach, which makes the different solvers much easier to implement and allows distributed computation. The major drawback of the standard partitioned FSI coupling scheme is that where higher density fluids are involved (meaning strong effects of added mass), convergence is no longer guaranteed, and divergence will generally be observed, regardless of the chosen time step for incompressible flows \cite{7}. A number of approaches have been proposed since the last decade to counter this drawback including semi-implicit discretization \cite{8} and adaptive Aitken under-relaxation \cite{9}, but convergence is not always guaranteed, or may be slow in cases of high-density fluids such as blood or water. Our objective in this paper is to show that in order to take into account heavy fluid flows such as in sea currents, the coupling scheme must be corrected, as described for example in \cite{1}, in order to counteract the penalizing impact of the added mass effect on the classical FSI coupling scheme. This correction is based on estimating an added mass matrix \([M_{add}]\) that may considerably improve and/or ensure the iterative phase of a strongly coupled partitioned approach. Moreover, the interest of an approach such as the Panel Method is here twofold, in that it enables this matrix to be estimated in a post-processing process without the use of a fluid mesh: this constitutes a serious and solid advantage.

2 MATHEMATICAL MODELS

2.1 Dynamics for a hydrofoil with two degrees of freedom

Here we consider a 2D airfoil with chord length \(c\), of mass \(m\) and flexibly attached to a fixed point, as illustrated in Figure 1. We have the pivot point \(P\) (also called the elastic axis), the aerodynamic center \(F\) (located at \(c/4\) from the leading edge) and the center of mass location \(G\). Two \(dof\) are here considered, namely plunging \(w(t)\) and a pitching \(\theta(t)\) motions. The vertical and rotational components of the airfoil velocity are respectively denoted by \(V_G\) and \(\theta\).

![Figure 1: Dynamics of a 2D hydrofoil cross section with two \(dof\) \(w(t)\) and \(\theta(t)\)](image)

The fundamental equations may be obtained from the kinetic and potential energy of the
airfoil:

\[
\mathcal{E}_k = \frac{1}{2} \left( m \mathbf{v}_G^2 + \mathcal{I}_G \dot{\theta}^2 \right) \quad \text{and} \quad \mathcal{E}_p = \frac{1}{2} \left( K_z \mathbf{w}^2 + K_\theta \dot{\theta}^2 \right) - P G m g \theta,
\]

with \( K_z \) denoting the axial rigidity along the \( z \)-axis, \( K_\theta \) the torsional rigidity with respect to the \( y \)-axis and \( \mathcal{I}_G \) the mass moment of inertia about \( G \). From Lagrange’s equation we then obtain the set of two equations:

\[
\begin{bmatrix}
m & -m P G \\
-m P G & I_P
\end{bmatrix}
\begin{bmatrix}
\ddot{w} \\
\ddot{\theta}
\end{bmatrix}
+ \begin{bmatrix}
K_z & 0 \\
0 & K_\theta
\end{bmatrix}
\begin{bmatrix}
w \\
\theta
\end{bmatrix}
= \begin{bmatrix}
\mathcal{R}_z \\
\mathcal{M}_P + P G m g
\end{bmatrix}
\tag{1}
\]

where \( \mathcal{R}_z \) and \( \mathcal{M}_P \) denote respectively the vertical component of the generalized force obtained from the pressure integration, and the resulting pitching moment at \( P \). Finally, this may be condensed to the following:

\[
[M] \{\ddot{U}\} + [K] \{U(t)\} = \{F_p(t)\}, \quad \text{with} \quad \{U(0)\} = \{U^0\} \quad \text{and} \quad \{\dot{U}(0)\} = \{0\},
\tag{2}
\]

where \( [M] \) and \( [K] \) denote respectively the mass and the rigidity matrices corresponding to the attachment of the airfoil, and \( \{U\} \) denotes the two dof. The term \( \{F_p\} \) denotes for the solicitation vector resulting from aerodynamic loads.

### 2.2 Panel Method (PM) for lifting potential flows

Panel methods are particularly suitable for calculating the flow field over an airfoil that undergoes unsteady time-dependent motion in a fluid that may be assumed inviscid and incompressible. Let \( \varphi \) define the total potential such that \( \mathbf{V} = \nabla \varphi \). Combining this with the first of the two equations just cited, we obtain the classical Poisson equation \( \Delta \varphi = 0 \), \( \forall \vec{x} \in \mathcal{V} \).

The main idea in the Panel Method is not to solve this Laplacian equation in the classical way for the entire fluid domain, but to cast the same analysis in a boundary integral equation form for which [10], where the Hess & Smith Panel Method (HSPM) was introduced, is considered to be the reference paper. In this approach, with 2D non-stationary flows being restricted as set out in [11], the velocity \( \mathbf{V} \) at any point \( \vec{x} = (x, z) \) of the fluid domain is decomposed according to:

\[
\mathbf{V}(\vec{x}) = \mathbf{V}_\infty + \vec{v} \quad \text{with} \quad \vec{v} = \int_s \frac{\sigma(s)\vec{r}}{2\pi r^2} ds + \int_s \frac{\tau(s)\vec{e}_\theta}{2\pi r} ds, \tag{3}
\]

where \( \mathbf{V}_\infty \) defines the velocity of the uniform flow at infinity. The vector \( \vec{v} \) denotes the disturbance field due to the airfoil and results from two contributions, since the airfoil may be represented by two elementary flows (also called singularities) corresponding to source flow \( (\sigma(s)) \) and vortex flow \( (\tau(s)) \). This expression satisfies the irrotationality condition and the boundary condition at infinity that stipulates the cancellation of the disturbance velocity, and it results from Green’s identity (we refer to [2] for a complete mathematical analysis). For a given set of source strength \( \sigma(s)ds \) and vortex strength \( \tau(s)ds \) (assumed uniform on the airfoil), it is then theoretically possible to define the velocity field at any point of the fluid domain: in the approach adopted in [10], the vortex strength \( \tau(s) \) is taken to be constant on the airfoil and adjusted to satisfy a lifting condition.
For points belonging to the interface between the flow and the airfoil, the boundary condition that stipulates no flow through surface enables us to define a given set of equations to be solved for \( \sigma(s) \). Completed by the Kutta condition that stipulates that the flow must leave the trailing edge smoothly, the set to be solved for \( \sigma(s) \) and \( \tau \) is now complete.

In order to couple the fluid flow with the structure, we need to know the pressure \( p \). This may be calculated at any point using the non-stationary form of Bernoulli’s equation [12].

\[
p + \rho gz + \rho \frac{V^2}{2} + \rho \frac{\partial \phi}{\partial t} = f(t).
\]

### 3 NUMERICAL MODELS

#### 3.1 Structure model

The time resolution of equation (2) is here obtained using a Newmark-Wilson finite difference [13] scheme such as:

\[
\left( \frac{4}{\Delta t^2}[M] + [K] \right) \{ \Delta U \} = \{ F_p \}^n - [K]\{U\}^n + [M] \left( \frac{4}{\Delta t} \{ \dot{U} \}^n + \frac{1}{4} \{ \ddot{U} \}^n \right).
\]

The indexes \( n \) and \( n + 1 \) correspond to the times \( t \) and \( t + \Delta t \) and \( \{ \Delta U \} = \{ U \}^{n+1} - \{ U \}^n \).

It should be pointed out that the fluid load term \( \{ F_p \} \), resulting from fluid pressure integration on the airfoil, is here computed at time step \( n \) because of the partitioned nature of the coupling scheme that we are considering. The mechanical energy can be calculated for each time step and is divided into two parts, respectively kinetic and potential:

\[
E_m = E_k + E_p \quad \text{with} \quad E_k = \frac{1}{2} < \dot{U} > [M]\{ \dot{U} \}, \quad E_p = \frac{1}{2} < U > [K]\{U\}.
\]

#### 3.2 Fluid model

The fluid flow numerical model is built in accordance with [11]. It first requires the discretization of the airfoil that is decomposed (see Figure 2) with \( N \) panels for \( N + 1 \) nodes (symbol \( \circ \)). Each panel \( i \) is defined by a pair of boundary points, \( (x_i, z_i) \) and \( (x_{i+1}, z_{i+1}) \), and a control point \( (cp_i) \) located at the midpoint (symbol \( \times \)).

![Figure 2: Panel representation of the airfoil surface](image)

The vectors \( \vec{n}_i \) and \( \vec{t}_i \) located at control point \( i \) denote respectively the normal and the tangential vectors. The perimeter of the airfoil is denoted as \( l \).
3.2.1 Stationary form

The fluid velocity at any point in the fluid domain is obtained by integrating equations (3) on the airfoil surface. This integration involves summing all panel contributions such that:

$$\vec{V}(P) = V_\infty \vec{r} + V_\infty \vec{k} + \vec{v}$$

where

$$\vec{v} = \sum_{j=1}^{N} \frac{\sigma_j \vec{r}}{2\pi r^2} ds + \sum_{j=1}^{N} \frac{\tau}{2\pi r} \vec{e}_\theta ds$$

(6)

For each control point \(i\), with the boundary condition of non-penetrating flow, integration yields the following expression:

$$(V_n)_i = V_\infty \vec{n}_i + \sum_{j=1}^{N} A_{nj} \sigma_j + \tau \sum_{j=1}^{N} B_{nj} = 0, \quad i = 1, \ldots, N$$

(7)

The terms \(A_{nj}\) and \(B_{nj}\), detailed in [11], are the influence coefficients for the source and vorticity distributions: the first subscript \(i\) denotes the panel that is subject to the influence of the panel indexed by \(j\).

In order to ensure the lifting capability of the airfoil, we here consider a Kutta-Joukowski condition stipulating that the magnitude of the two tangential velocities at the trailing edge (control point 1 and \(N\)) must be equal to each other:

$$(V_t)_N = -(V_t)_1 \quad \text{with} \quad (V_t)_i = V_\infty \vec{t}_i + \sum_{j=1}^{N} A_{tij} \sigma_j + \tau \sum_{j=1}^{N} B_{tij},$$

(8)

The minus sign results from the fact that the two tangential vectors \(\vec{t}_1\) and \(\vec{t}_N\) are opposite. The overall system that needs to be solved comes out of equations (7) and (8), and may be written:

$$\begin{bmatrix}
A_{t1}^n & \cdots & A_{tN}^n \\
\vdots & \ddots & \vdots \\
A_{t1}^t + A_{tN1}^t & \cdots & A_{t1}^t + A_{tN1}^t \\
\end{bmatrix}
\begin{bmatrix}
\sigma_1 \\
\vdots \\
\sigma_N \\
\tau
\end{bmatrix} =
\begin{bmatrix}
-V_\infty \vec{n}_i \\
\vdots \\
-V_\infty \vec{t}_N
\end{bmatrix}$$

Solving this system allows the pressure to be computed at all control points, using the following:

$$p_i = \rho \frac{V_\infty^2}{2} C_{pi} \quad \text{with} \quad C_{pi} = 1 - \left(\frac{(V_t)_i}{V_\infty}\right)^2,$$

(9)

where \(C_{pi}\) denotes the nodal pressure coefficient.

3.3 Non-stationary form

According to Kelvin’s circulation theorem [12], the circulation \(\Gamma\) around a closed curve \(C_1\) (composed of the same fluid particles) moving with the fluid, remains constant with time. Stated mathematically:

$$\Gamma_{C_1|t=0} = \oint_{C_1} \vec{V} \cdot d\vec{s} = 0 \Rightarrow \frac{d\Gamma_{C_1}}{dt} = 0, \forall t \geq 0$$
The lifting process around an airfoil is directly related to a circulation, in accordance with the Kutta-Joukowsky theorem. We deduce that any changes in the airfoil circulation must be balanced by an equal and opposite change in the wake, resulting from the vortex shedding at the trailing edge of the airfoil.

At time step \((n+1)\), the vortex shedding has a corresponding additional wake element indexed by \(w\), of uniform vorticity \(\tau_{w}^{n+1}\) and length \(\Delta x_{n+1}\) such that:

\[
\Theta^{n+1} = \tan^{-1}\left(\frac{u_{w}^{n+1}}{v_{w}^{n+1}}\right), \quad \Delta x^{n+1} = \Delta t \sqrt{(u_{w}^{n+1})^2 + (v_{w}^{n+1})^2},
\]

where \((u_{w}, v_{w})\) denote the velocity components at the trailing edge and

\[
\tau_{w}^{n+1} \Delta x_{n+1} = \Gamma^{n} - \Gamma^{n+1} = l(\tau^{n} - \tau^{n+1}).
\]

Each time step \(n\) is then associated with a detached vortex \(n\) of circulation \(\Gamma^{n} = \tau_{w}^{n} l\) convected downstream (see Figure 3) by the fluid velocity field.

The non-stationary form of the original equation (7) is then completed by unsteady effects and rearranged using the terms from equation (11) such that:

\[
\sum_{j=1}^{N} A_{ij}^{n} \sigma_{i}^{n+1} = \tau^{n+1} \left(\frac{l}{\Delta x_{n+1}} (B_{iw}^{n})^{n+1} - \sum_{j=1}^{N} B_{ij}^{n}\right) - (\vec{V}_{ST}.\vec{n})^{n+1} \\
- \tau^{n} \frac{l}{\Delta x_{n+1}} (B_{iw}^{n})^{n+1} - \sum_{m=1}^{n} (C_{im}^{n})^{n} (\Gamma^{m-1} - \Gamma^{m}).
\]

The term \(\vec{V}_{ST}\) denotes the unsteady upstream velocity seen from a coordinate system attached to the airfoil in order to carry the airfoil motion [11] over to the relative fluid velocity calculation such as:

\[
\vec{V}_{ST} = \vec{V}_{\infty} - \dot{w}\hat{k} + \dot{\theta}\hat{j} \times (\vec{r} - \vec{r}_{P}),
\]

with \(\dot{w}\) the vertical velocity component of the airfoil, \(\dot{\theta}\) its rotational velocity and \(P\) its rotation axis (see Figure 1). The resulting algebraic equations can be written in the form:

\[
[A]^{n+1} \{\sigma\} = \tau^{n+1} \{B\} + \{C\}.
\]

Vectors \(\{B\}\) and \(\{C\}\) directly depend on the shed vorticity panel values \((\Delta x^{n+1}, \Theta^{n+1}, \tau_{w}^{n+1})\) and consequently an iterative procedure must be set up to solve the system (14).
3.4 Incorporating added mass effects into the fluid-structure coupling scheme

The coupling process is required to perform FSI calculations in order to regularly update the variables common to both the fluid and the structure solvers. The exchange process should be read as follows:

1. Starting at time $n$, a single time step is executed in order for the structure solver to update displacements and velocities, bringing us to time $n + 1$.

2. The information based on the new structure state is transferred to the fluid code.

3. Starting at time $n$, a single time step is executed to update all fluid data, bringing us to time $n + 1$.

4. The pressure field is transferred to the structure code.

... Steps [1→4] are executed iteratively until some convergence criterion is satisfied.

In order to better counteract the added mass effect that results from heavy fluid flow such as in a liquid (sea currents) and that may lead to divergence, here we propose correcting the classical FSI coupling scheme in relation to the added mass effect. The main idea (in the case of conservative systems only) is that if the real added mass matrix $[M_{add,f}]$ could be calculated exactly, the force term appearing in equation (2) would be exactly replaced by:

$$\{F_p\}_i \equiv -[M_{add,f}] \{\ddot{U}\}_i.$$  (15)

For most cases, the real added mass matrix $[M_{add,f}]$ is out of reach. The classical partitioned coupling scheme (denoted by CLAS) is then modified in accordance with [1], and equation (2) is now related to the corrected scheme (denoted by CORR):

$$([M] + [M_{add,e}]) \{\ddot{U}\}_{i+1} + [K] \{U\}_{i+1} = \{F_p\}_i + [M_{add,e}] \{\ddot{U}\}_i,$$  (16)

where $i$ and $i + 1$ are indexes for the iterative process, and $[M_{add,e}]$ is the matrix corresponding to the estimated added mass effect resulting from the pressure load all around the structure. Each component $M_{add,e}(i, j)$ is related to the force on the body in the $i$-axis resulting from a unit acceleration along the $j$-axis. At convergence, the two additional terms cancel out and we get back to the original form of the coupling equation (2). Adding extra terms on both parts of the original equation, in accordance with equation (15), helps to reduce the penalizing effect of $\{F_p\}$ and to increase the beneficial effect of $[M]\{\ddot{U}\}$ on the convergence process. The added mass matrix calculation is calculated according to [14]:

$$M_{add,e}(j, k) = \rho \iint_V \phi_{i,k} \phi_{j,k} dV = \rho \iint_S \phi_{i} \frac{\partial \phi_{j}}{\partial n} dS.$$  (17)

The above expression is fully compatible with the Panel Method in computing $\phi_{j}$, the gradient term simply being equal to the normal component of the parietal velocity of the body (a variable that is already known, transmitted by the structure solver).
4 RESULTS

4.1 Thrust generation by a forced oscillating airfoil

This first example is simply intended as a qualitative, graphical illustration of the capability of the Panel Method in relation to the well-known effect of thrust generation through airfoil oscillation (as presented in [11]). The same non-symmetric NACA 2412 airfoil is moved along its plunging and pitching directions, according to:

\[ w(t) = w_o \sin(\omega t), \quad \theta(t) = \theta_o \sin(\omega t + \phi) \]

where \( \phi \) introduces a phase shift between the two variables. The unsteadiness in the flow is defined by the reduced frequency \( \kappa = \frac{\omega c}{V_\infty} \): two different non-stationary calculations are conducted respectively for \( \kappa = 0.5 \) and \( \kappa = 2 \). The entire simulation takes place over eight periods of oscillation with 200 time steps per period. The airfoil (chord unity) is placed in a flow with \( V_\infty = +5 \text{ m/s} \), \( w_o = 0.2 \text{ m} \), \( \theta_o = 8^\circ \) and \( \phi = 60^\circ \) and the airfoil is decomposed into 105 panels.

![Figure 4: Different flow characterizations according to the reduced frequency \( \kappa \)](image)

Results of the two calculations are illustrated in Figure 4. In each case a wake appears that results from the successive generation of vortex shedding at the trailing edge of the airfoil. For different stations \( (x = 3, \ldots, 11 \text{ m}) \) downstream of the trailing edge, the time-averaged velocity profile (reduced from the value \( u_\infty \)) is superposed. The two cases clearly show opposing behaviors: in the first case (\( \kappa = 0.5 \)), the wake contributes to the airfoil drag (reverse flow), whereas the second case (\( \kappa = 2 \)) there is a jet effect in the direction of flow that can be used profitably to propel the airfoil.

4.2 Free coupling regime

In this third example the same airfoil (NACA 2412, 105 panels) is now flexibly attached to a fixed point, as illustrated in Figure 1. Immersed in a uniform flow \( (V_\infty, \rho) \), the airfoil is initially
removed from its position at rest \((w_0 = 0.2 \, m\) and \(\theta_0 = 8 \, deg\)) until a stationary fluid state is reached. It is then relaxed to allow the fluid-structure coupling process to take place freely: it will be remarked that the mechanical energy decreases with time because of a transfer of energy to the fluid tracked in the form of a vortex wake. The point of this example is to look at the influence of the volumetric mass \(\rho\) on the convergence property of the FSI scheme, to show the severe limit observed for the classical coupling scheme (that we term CLAS), and finally to show the beneficial effect of the scheme corrected from the added mass effect (that we term CORR).

Fluid flow conditions and structure characteristics are summarized in Table 1.

| \(\rho\) \([\, kg/m^3]\) | \(V_\infty\) \([m/s]\) | \(m\) \([kg]\) | \(I|_G\) \([kg \cdot m^2]\) | \(K_Z\) \([N/m]\) | \(K_\theta\) \([Nm/rad]\) |
|---|---|---|---|---|---|
| \([1 - 2000]\) | 5 | 10 | 100 | \(10^4\) | \(10^4\) |

Table 1: Fluid flow conditions and structure properties

The estimated added mass matrix \([M_{add,e}]\) is calculated with equation (17) to obtain:

\[
M_{add,e}(1,1) = 0.71 \rho, \quad M_{add,e}(1,2) = M_{add,e}(2,1) = -0.15 \rho, \quad M_{add,e}(2,2) = 0.05 \rho.
\]

The airfoil is considered fixed over a given number of time steps \(n_{FIX}\), then its flexibility is restored in order to start the free fluid-structure interaction. The time step \(\Delta t\) is related to the two natural frequencies, extracted from an eigenvalue analysis:

\[
f_1 = \frac{1}{\tau_1} = 1.59 \, Hz, \quad f_2 = \frac{1}{\tau_2} = 5.03 \, Hz \quad \text{such as} \quad \Delta t = \frac{\min(\tau_1, \tau_2)}{200} \approx 10^{-3} \, s.
\]

The same analysis, in accordance with equation (16), is conducted over \(n_{step}\) time steps, for both the CLAS scheme (with \([M_{add,e}] = [0]\)) and the CORR scheme, for a range of volumetric mass given in Table 1. For each time step, the number of iterations to convergence is extracted. Figure 5 illustrates the two calculations for the limit case \(\rho = 21 \, kg/m^3\). The upper graph shows the different wake patterns: black dots for CORR and larger, colored symbols for CLAS. With CLAS the wake pattern is seen to be wider, and the total number of iterations for convergence (lower graph) is systematically higher than with CORR.

This case corresponds to the limit observed for the CLAS scheme to converge. Any density above this value causes the coupling to diverge, as reported in Table 2. For the considered case, the CLAS scheme excludes densities higher than 20 \(kg/m^3\). Above this value the coupling scheme systematically diverges. The CORR scheme, on the other hand, systematically converges, whatever the density value, which confirms its capacity to support heavy fluids. But the number of iterations is not the only determinant of whether a scheme is capable of converging. If the aerodynamic coefficients are plotted with respect to time, it can clearly be seen that even where convergence is reached, the solution may not be physically acceptable. For \(\rho \in [8, 20] \, kg/m^3\) (not
shown here) is can be observed (in the case of the CLAS coupling scheme) that all aerodynamic coefficients oscillate and, second, that the drag coefficient $C_D$ that ought to be null is in fact non-zero!

![Convergence histories for CLAS and CORR schemes](image)

Figure 5: Convergence histories for CLAS and CORR schemes (case $\rho = 21 \, \text{kg/m}^3$)

The results below are solely obtained by using the CORR scheme. Figure 6(a) and (b) shows that increasing the fluid density gives rise to a predictable higher level of energy dissipation in the fluid, since a wake and vortex are generated that are simply convected downstream of the flow with their own energy (which the airfoil is consequently deprived of). Energy signals are normalized according to the initial energy $E_o$ resulting from initial perturbations $w_o$ and $\theta_o$.

The two signals $E_m$ and $W$ have opposite behaviors, in agreement with the principle of energy conservation. In other words, an irreversible transfer of the mechanical energy $E_m$ (equation 5) is observed between the main flow and the airfoil, due to the produced work $W$. The higher the density, the higher the observed dissipative effect.
Figure 7 illustrates the frequency shifting observed and predicted for the two modes (pitching and plunging), directly related to the density value \( \rho \) (in log-scale). Theoretical predictions (solid lines), based on an eigenvalue analysis (mass, added mass and rigidity matrices), are compared with the observed frequencies extracted (symbols) from the FSI calculations after an FFT analysis.

![Figure 7: Modal frequencies with respect to \( \rho \): predictions (lines) and FSI calculations (symbols)](image)

It will be noted that the pitching frequency is relatively constant, whereas the plunging mode is quite sensitive to the value of the density. The eigenvalue analysis predicts a modal coalescing at the critical value \( \rho_{\text{coal}} = 133 \text{ kg/m}^3 \). The FSI simulations confirm the coalescing effect, but for \( \rho = 50 \text{ kg/m}^3 \).

5 Conclusions and prospects

This paper presents a corrected version of a strongly iterative partitioned FSI scheme for studying the dynamics of an airfoil flexibly attached and immersed in a heavy fluid. The intended application of our work mainly concerns the fluid-structure coupling that may operate between a moving lifting component (such as a marine current turbine) and a surrounding heavy fluid such as water. The mathematical model for the fluid is based on the potential Panel Method that offers the dual benefit of being restricted to a boundary element analysis and of ensuring the lifting capability of the component. The mathematical model for the structure, on the other hand, is based on the dynamics of a 2D airfoil that encounters plunging and pitching motions. We demonstrate that in simple case, the iterative convergence of a classical FSI partitioned scheme ceases to be guaranteed once the added mass exceeds the mass of the component. Correcting the FSI scheme to counteract the penalizing effect of the added mass allows convergence to be ensured, whatever the value of the added mass. The first example, for a forced oscillating airfoil, validates the fluid flow model for a stationary flow. The second example is intended to show how the classical FSI scheme is only applicable to a narrow range of fluids (\( \rho \leq 8 \text{ kg/m}^3 \)), whereas taking into account the added mass effect on the coupling scheme can ensure the convergence required by coupling considerations. A coalescing and stable effect were observed, which also illustrates that the natural modes resulting from the flexibility of the structure may vary considerably. However, even though taking an estimated added mass matrix into account has obvious benefits in relation to FSI coupling in heavy fluids, this alone is not sufficient for accurately estimating the consequences of energy transfers that significantly modify the energy absorbed or dissipated by the system. We are currently looking at the possibility of
extending this approach to 3D applications to cover more realistic rotor geometries (wind mills, marine turbines), in order to establish the full requirements of FSI calculations for such devices.

REFERENCES


NUMERICAL PREDICTION FOR MANY FLOATING DEBRIS TRANSPORTED IN CITY MODEL DUE TO TSUNAMI-INDUCED FLOWS

S. USHIJIMA*, D. TORIU*, K. AOKI**, H. ITADA† and D. YAGYU‡

* ACCMS, Kyoto University, Sakyou-ku, Kyoto-city, 606-8501, Japan
e-mail: ushijima.satoru.3c@kyoto-u.ac.jp

** IHI Corporation, 1-1, Toyosu 3, Koto-ku, Tokyo, 135-8710, Japan

† Mitsui O.S.K. Lines, Ltd., 1-1 Toranomon 2-chome, Minato-ku, Tokyo, 105-8688, Japan

‡ CERE, Kyoto University, Katsura, Kyoto-city, 615-8540, Japan

Key words: Tsunami, Floating Debris, Multiphase Model, Parallel Computation

Abstract. A three-dimensional computational method based on multiphase modelling is employed to predict the behaviors of floating tsunami debris in coastal residential areas. The present computational method enables us to deal with the interactions between free-surface flows and the movements of floating objects, as well as the collisions among the objects and fixed structures. The present method was first applied to simple stability problems of floating cylinders and then it was applied to the 1/250 scale tsunami experiments. Finally, two types of numerical experiments were performed using larger number of floating objects in more complicated conditions. As a result, it was shown that the present method is effective to predict the behaviors of floating objects transported by tsunami between buildings on non-uniform ground surfaces.

1 INTRODUCTION

In the Great East Japan Earthquake in 2011, floating debris transported by tsunami caused serious secondary damage against buildings and other structures in coastal residential areas. Thus, it is important to develop a numerical method to estimate the behaviors and distributions of the floating objects transported by tsunami in coastal regions.

There have been some studies to propose computational methods for solid objects drifting in waves and tsunami as reported by Kawasaki et al. [1], Yoneyama et al. [2] and others. In contrast to the usual methods, a large number of floating objects can be calculated accurately with the present method in more complicated conditions, since it gives us the reasonable solutions for the difficulties to treat the collisions among objects, fluid-solid interactions and computational load to calculate large problems.
In this study, a computational method based on a multiphase modelling (called MICS) \[3\] is employed and the outline of the governing equations and numerical procedures is shown, in order to calculate floating objects transported by tsunami flows, taking account of the fluid-solid interactions and collisions among the objects and fixed structures.

The basic validity of the present method was first confirmed by applying it to the stability problems of floating cylinders. Then it was applied to the experimental results obtained in a 1/250 scale city model in the flume of DPRI in Kyoto University. As a result of the calculations, it was shown that the predicted distributions of 42 floating objects transported in the city model are in good agreement with the experimental results. On the basis of the above validation, two types of numerical experiments were conducted: (1) 240 truck models transported by dam-break flows with and without debris control structures (DCS) on non-uniform ground surfaces and (2) 156 floating objects drifting in a 1/250 scale city model whose area is larger than the experimental one. As a result of the numerical experiments, it was demonstrated that the present computational method is expected to enable us to propose various prevention measures and effective design against the secondary disaster caused by tsunami debris.

2 NUMERICAL PROCEDURES

2.1 Multiphase model and basic equations

In the numerical prediction of tsunami debris, it is necessary to use the advanced computational method that enables us to calculate three-dimensional free-surface flows and fluid-solid interactions as well as the collisions among solid objects. For that purpose, three-dimensional parallel computation method (MICS) \[3\], which has been proposed on the basis of the multiphase model, is employed in this study. Figure 1 (a) illustrates the multiphase field, consisting of gas, liquid and solid phases, in a computational cell. The present model was derived by assuming that all phases are incompressible and immiscible with the averaging procedures for multiple phases \[4\]. The governing equations for gas-liquid phase consist of the incompressible condition and equations for mass and momentum conservations, which are given as follows:

\[
\frac{\partial u_i}{\partial x_i} = 0 \quad (1)
\]

\[
\frac{\partial \rho_f}{\partial t} + \frac{\partial (\rho_f u_i)}{\partial x_i} = 0 \quad (2)
\]

\[
\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{1}{\rho_f} \frac{\partial p}{\partial x_i} + g_i + \frac{1}{\rho_f} \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] \quad (3)
\]

where \( t \) is time, \( x_i \) is the component of the orthogonal coordinate system and \( g_i \) is the \( x_i \) component of the external acceleration. While \( u_i \) is the mass-averaged velocity of gas and liquid phases, the volume-averaged variables are used for pressure \( p \), density \( \rho_f \) and coefficient of viscosity \( \mu \).
2.2 Outline of numerical procedures

Equations (1), (2) and (3) are discretized with a finite volume method on the collocated grid system, which is a structured Eulerian grid system. In contrast, the solid objects are represented by multiple tetrahedron elements as schematically shown in Fig. 1 (b), whose node points are treated in a Lagrangian way on the Eulerian grid system used for gas and liquid phases.

The numerical solutions for gas and liquid phases are calculated with the modified SMAC method from the original one [6], in which an implicit discretization technique, a C-ISMAC method [7], is applied to reduce elapsed time for computations in addition to easily implement the fifth-order TVD scheme [8] for convection terms in Eqs. (2) and (3). In the calculations of a pressure field, a C-HSMAC method [3] is employed to satisfy incompressible condition. As a result of the C-HSMAC method, the discretized velocity components defined on cell boundaries, which are used to calculate fluxes in the finite volume method, satisfy accurately the incompressible condition given by Eq. (1).

In addition, in an attempt to reduce the elapsed time of computation, the present method is completely parallelized with flat MPI (Message-Passing Interface) [9] on the basis of a three-dimensional domain decomposition method.

2.3 Calculations of floating objects

The floating objects transported by tsunami flows are assumed to be rigid bodies which have no deformation due to the collision with the other objects, while the contact forces between objects are estimated using multiple contact-detection spheres set up within the solid model as shown in Fig. 1 (b). The normal and tangential forces arising in the contact are calculated on the basis of a distinct element method (DEM) [10].

The movements of the floating objects are calculated with the basic equations for translational
and rotational motions. The basic equation of the translational motion is given by

\[ M_b \ddot{v} = F_F + F_C \quad (4) \]

where \( M_b \) is the mass of an object, \( \dot{v} \) is the velocity vector of its center and dot means time differentiation. On the right hand side of Eq. (4), both fluid force \( F_F \) and contact force \( F_C \) are taken into account. The basic equation of the rotational motion, Euler equation, is given by

\[ \dot{\omega} = I^{-1} \left[ R^{-1} N - \omega \times I \omega \right] \quad (5) \]

where \( \omega \) is the angular velocity vector, \( I \) is inertia tensor in the basic attitude of the object, \( R \) is a rotation matrix and \( N \) is the external torque resulting from \( F_F \) and \( F_C \). The calculation related to the rotation is actually performed with quaternion instead of the multiplication of the rotation matrix \( R \). The location and attitude of the floating object are determined from Eqs.(4) and (5). The \( x_i \) component of the fluid force \( f_{Fi} \), which acts on a part of an object included in one cell, is calculated with the pressure and viscous terms of Eq. (3):

\[ f_{Fi} = \Delta m \left[ -\frac{\rho_b - \rho_f}{\rho_b} g \delta_{3i} - \frac{1}{\rho_b} \frac{\partial p}{\partial x_i} + \frac{1}{\rho_b} \frac{\partial}{\partial x_j} \left\{ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right\} \right] \quad (6) \]

where \( \Delta m \) is the mass of the part of the object included in a cell and \( \rho_b \) is the volume-average density including solid phase in a cell. Taking \( x_3 \) axis vertically upward, \( g \) is the acceleration of gravity and \( \delta_{3i} \) is a Kronecker delta. The summation of \( f_{Fi} \) over all cells including the same object corresponds to the component of the fluid force \( F_F \) on the right hand side of Eq. (4). The volume fraction of solid phases included in a computational cell, such as \( \Omega_1 \) illustrated in Fig. 1 (a), is estimated with a sub-cell method [5].

3 VALIDATION OF COMPUTATIONAL METHOD

3.1 Stabilities of floating cylinders

It is important to estimate the stability and attitude accurately in the prediction of floating objects in free-surface flows. As a basic validity, the present method was applied to a simple problem, in which the attitudes of two floating cylinders classified into stable and unstable conditions are calculated with the present computational method. Figure 2 shows the geometry and variables. The specific gravity is 0.5 and the diameter \( D \) is 0.1 [m] for both cylinders. The axial lengths \( L \) are set as follows: \( L = 0.05 \) [m] in Cylinder-A (stable) and \( L = 0.1 \) [m] Cylinder-B (unstable). The stability of the cylinders due to initial disturbance is classified with the plus or minus sign of the following \( s \) :

\[ s = \frac{I_y}{V_s} - \frac{GB}{V_s} \quad (7) \]

where \( G \) and \( B \) are centers of gravity and buoyancy, while \( V_s \) is the immersed volume and \( I_y \) is the moment of inertia of the shaded area on waterline section, as shown in Fig. 2 (b), around \( y \)-axis.
The predicted results of two cylinders are shown in Figs. 3 and 4. The cylinders are represented by tetrahedron elements and the interactions between cylinders and the flows of air and water are also calculated. The initial disturbances $\theta$ indicated in Fig. 2 (a) are set at $\pi/6$ for Cylinder-A and $\pi/60$ for Cylinder-B. As shown in Figs. 3 and 4, it was confirmed that Cylinder-A stably returns to the initial attitude, while Cylinder-B is unstable as expected in theory.

**Figure 2:** Floating cylinder and coordinates

**Figure 3:** Calculated results of Cylinder-A (stable, $L/D = 0.5$)

**Figure 4:** Calculated results of Cylinder-B (unstable, $L/D = 1.0$)
3.2 Comparisons with tsunami experiments

Hydraulic experiments were conducted using Hybrid Tsunami Open Flume in Ujigawa in DPRI of Kyoto University, in an attempt to measure the transportations and distributions of floating objects in a 1/250 scale city model. Figure 5 (a) shows a schematic view of the flume, while Fig. 5 (b) is a photograph of a part of the city model including calculation area. In experiments, 42 rectangular floating objects, $10 \times 20 \times 10$ [mm] and density 893 [kg/m$^3$], are placed in front of the city model as shown in Fig. 5 (b). The inflow water of the flow rate $6.7 \times 10^{-2}$ [m$^3$/s] was provided during 60 [s], which causes the flows corresponding to actual tsunami about 5 [m] in wave height.

![Experimental flume and city model](image1)

**Figure 5:** Experimental conditions

The computational domain shown in Fig. 6 (a) is a part of the city model as indicated in Fig. 5 (b). The 42 floating objects and main buildings are included in the computational domain, $x_1 \times x_2 = 0.25 \times 0.95$ [m]. Figure 6 (b) shows the details of a computational floating model, which is represented by 134 ($\equiv N_e$) tetrahedron elements and 72 contact detection spheres. The density of the floating model is same as the experimental one. The number of computational cells is $350 \times 270 \times 250$ in $x_1 \times x_2 \times x_3$ directions and 300-parallel computations were executed.

![City model and floating model](image2)

**Figure 6:** Conditions of computations
Figure 7 shows the distributions of the center points of the floating objects, which were measured three times in experiments. It was shown that the floating objects are transported through the wide streets between buildings with the passage of time. The calculated distributions of the floating objects are shown in Fig. 8. In calculations, the floating objects shown in Fig. 6 (b) are transported by the free-surface flows colliding with the other objects and buildings. It was confirmed that the calculated distributions of the objects are almost in good agreement with the experiments from the comparisons between Figs. 7 and 8.

![Figure 7](image1.png)

(a) $t = 0.0 \text{ [s]}$  
(b) $t = 3.0 \text{ [s]}$  
(c) $t = 6.0 \text{ [s]}$

**Figure 7**: Experimental results of floating objects transported in city model (3 cases)

![Figure 8](image2.png)

(a) $t = 0.0 \text{ [s]}$  
(b) $t = 3.0 \text{ [s]}$  
(c) $t = 6.0 \text{ [s]}$

**Figure 8**: Calculated results of floating objects transported in city model

4 NUMERICAL EXPERIMENTS

4.1 Floating trucks and debris control structures (DCS)

Numerical experiments were conducted with the present method in order to confirm its applicability to more complicated-shaped floating objects and non-uniform ground surfaces in addition to estimating the effects of debris control structures (DCS) [11].
Figure 9 shows the conditions of calculations, which correspond to a small scale model. As shown in Fig. 9 (a), in a computational domain, $l_1 \times l_2 \times l_3 = 2.0 \times 1.0 \times 0.2$ [m], rectangular water mass is initially placed in the area, where $x_1 \leq 0.3 \ l_3$ and $x_3 \leq 0.4 \ l_3$, to cause dam-break flow in the domain. Figure 9 (b) shows 240 floating truck models and 10 fixed cylindrical DCS with grand surface which has two hemisphere-like high areas. The diameter of DCS is 90 [mm]. The shape of each truck model is shown in Fig. 9 (c), which is represented by tetrahedron elements, whose number $N_e = 399$, as indicated in Fig. 9 (d) with wire-frames. The densities of DCS and truck models are $3.57 \times 10^2$ and $2.56 \times 10^3$ [kg/m$^3$] respectively.

In the calculations, the number of computational cells is $400 \times 200 \times 40$, while the domain is decomposed into $20 \times 10 \times 2$ with 400-parallel computations. The time increment $\Delta t$ is $5.0 \times 10^{-4}$ [s]. Figures 10 and 11 show the predicted results for two cases of computations: with DCS and without DCS. As shown in Fig. 10, without DCS, it was seen that the truck models are transported far into the downstream area on the lower grand surface due to the free-surface flows. In contrary, as shown in Fig. 11, it was confirmed that most of the truck models are trapped in front of DCS. From the above results, it can be concluded that the present method is expected to provide us with the efficient measures and design methods against the secondary disaster of tsunami, which is caused by many floating objects.
4.2 Tsunami debris transported in a wide range of city model

Numerical experiments were conducted with a 1/250 scale artificial city model whose area is larger than the experiments described in 3.2, in order to confirm the applicability of the present method to more complicated city model with many floating objects.

Figure 12 (a) shows the conditions of computations. The flow rate from the inlet area is $1.353 \times 10^{-3} \text{ m}^3/\text{s}$. The 156 floating objects, which assume to be nearly maximum-loading containers, were initially placed in front of the city model as shown in Fig. 12 (b). Their geometry is rectangular $10 \times 20 \times 10 \text{ [mm]}$ and the density is $600 \text{ [kg/m}^3\text{]}. Each floating object is represented by 134 tetrahedron elements and each includes 16 contact-detection spheres.

The number of computational cells was $345 \times 231 \times 64$ and the computational domain in Fig. 12 (a) was decomposed into $15 \times 11 \times 2$ subdomains, to which 330-parallel computation was applied. The total elapsed time was about 59 hours to calculate $24.0 \text{ [s]}$ shown in Fig. 12 (d).

As shown in Figs. 12 (b) to (d), the floating objects are transported by the free-surface flows colliding with the other objects and buildings. It is seen that they move through relatively wide spaces between buildings and that the transition of their distributions is reasonably predicted in the flooded regions.
5 CONCLUSIONS

In this study, a computational method based on a multiphase modelling (MICS) [3] was employed and its governing equations and numerical procedures were shown in order to calculate the floating objects transported by tsunami flows taking account of the fluid-solid interactions and collisions among many objects and fixed structures.

The basic validity of the present method was first confirmed by applying it to the stability problems of floating cylinders. Then it was applied to the experimental results obtained in a 1/250 scale city model in the flume of DPRI in Kyoto University. As a result of calculations, it was shown that the predicted distributions of 42 floating objects transported in the city model are in good agreement with the experimental results. On the basis of the above validation, two types of numerical experiments were conducted: (1) 240 truck models transported by dam-break flows with and without debris control structures (DCS) on non-uniform ground surfaces and (2) 156 floating objects in a 1/250 scale city model whose area is larger than the experimental one. As a result of the numerical experiments, it was demonstrated that the present computational method is expected to enable us to propose various prevention measures and design against the secondary disaster of tsunami.

Figure 12: Conditions of calculations and predicted results
ACKNOWLEDGEMENTS

The experiments in DPRI were supported by Prof. N. Yoneyama and Prof. N. Mori in Kyoto University. The authors would like to thank them for their contribution to this study.

REFERENCES


SIMULATION OF BREAKING FOCUSED WAVES OVER A SLOPE WITH A CFD BASED NUMERICAL WAVE TANK

MAYILVAHANAN ALAGAN CHELLA, HANS BIHS, CSABA PÁKOZDI, DAG MYRHAUG AND ØIVIND ASGEIR ARNTSEN

1 Department of Civil and Environmental Engineering, NTNU
Trondheim, Norway
2 SINTEF Ocean
3 Department of Marine Technology, NTNU
Trondheim, Norway

Key words: Numerical simulation, Focused waves, Breaking waves, Single flap maker, Breaking waves

Abstract. Extreme wave conditions are always identified with large-amplitude breaking waves in shallow waters. Focused waves can often be used to describe extreme waves which evolve during the nonlinear wave-wave interaction, occurring at one point in space and time. Understanding breaking focused waves has many design-related implications for the design of offshore wind turbine (OWT) substructures in shallow waters. The main objective of the paper is to model breaking focused waves over a sloping seabed and study the breaking characteristics using the open-source CFD model REEF3D. The numerical model describes the two-phase flow using the incompressible Reynolds-Averaged Navier-Stokes (RANS) equations together with the continuity equation. The model uses a fifth-order WENO scheme for convection discretization and a third order Runge-Kutta scheme for time discretization along with the level set method to obtain the free surface, yielding accurate wave propagation in the numerical wave tank. Solid boundaries are accounted through the ghost cell immersed boundary method. The free surface is modeled with the level set method. Turbulence is described with the two-equation $k-\omega$ model. In the numerical wave tank, the focused waves are generated using a single flap-type maker theory. The numerical results are in good agreement with experimental results for complex free surface elevations measured at several locations along the wave tank. The numerical aspects related to the development of the breaking process are investigated together with the evolution of focusing wave group in the numerical wave tank. Further, the study also examines the free surface flow features that evolve during the breaking process.

1 INTRODUCTION

Hydrodynamic loads from extreme waves on offshore wind turbine substructures is an important design criterion [1]. Extreme waves are characterized by a single large steep wave crest
with high degree of asymmetry. They are much larger than expected for the normal sea state. Despite that the extreme events are exceptional events, they can cause severe damage to offshore structures. Considering the nonlinear behavior of shallow water waves, the shortening of wave length resulting in larger wave heights. In addition to the strong wave-wave interactions, the waves are affected by the seabed bathymetry. This causes the drastic changes in the wave transformation characteristics. The extreme waves events can be modelled with focused waves which are results of superposition of many linear wave components concentrating at the intended space and time.

Waves grow steeper and higher and they break more frequently in shallow waters. The interaction of breaking waves with offshore structures exerts severe hydrodynamic loading on them.

There have been limited studies on these engineering aspects relevant to the hydrodynamic load assessment parameters in shallow waters, especially for breaking focused waves over a sloping seabed. Despite several numerical studies based on computational fluid dynamics (CFD) have been carried out to model non-breaking focused waves over constant depth [2, 3, 4, 5, 6, 7], there have been limited numerical studies on focused waves over slopes, particularly breaking focused waves. In addition, the breaking characteristics of focused waves over a slope are not yet fully understood due to many parameters involved in the physical processes. It is quiet challenging to describe the evolution of a focused wave group over a slope since the focusing mechanism needs to be defined along with other physical processes such as shoaling and breaking.

The main purpose of the present paper is to simulate breaking focused waves over a slope and investigate the breaking characteristics with the open-source CFD model REEF3D [8]. First, the numerical model is validated by comparing the computed results with experimental data for wave surface elevations along the wave tank. A good agreement between the numerical results and experimental data is obtained. Further, the numerical aspects related to the development of the breaking process are investigated together with the evolution of focusing wave group in the numerical wave tank. The prominent free surface flow features evolve during the breaking process is also presented and discussed.

2 NUMERICAL MODEL

In the open-source CFD model REEF3D, the unsteady Reynolds-Averaged Navier Stokes (URANS) equations along with the continuity equation are solved for incompressible two-phase flow in the numerical wave tank[8]. The mass and momentum equations are:

\[
\frac{\partial u_i}{\partial x_i} = 0 \quad (1)
\]

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_t) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + g_i \quad (2)
\]

Here, \( u \) is the mean velocity components, \( p \) is the pressure, \( \rho \) is the fluid density, \( \nu \) is the kinematic viscosity, \( \nu_t \) is the eddy viscosity, \( i \) and \( j \) denote the indices in \( x \) and \( y \) direction, respectively and \( g_i \) is the gravity term. The discretization of convective terms of the URANS equations are treated with the fifth-order weighted essentially non-oscillatory (WENO) scheme [9] which is a higher order conservative finite difference scheme. For the time integration, the third-order accurate TVD Runge-Kutta scheme [10] is implemented. The time step determination is based on an
Figure 1: A definition sketch of flap wave maker

adaptive time stepping method [11] which is controlled by the Courant-Friedrichs-Lewy (CFL) criterion. The CFL criterion is maintained and the simulation time step is adjusted for each iteration. This method accounts for the effects of velocity and the source term $S$ on the temporal numerical solutions. Moreover, this approach enhances the robustness of the numerical model, the overall efficiency and the credibility of simulation results. The pressure term in URANS equation is modeled with the Chorins projection method [12]. First, the intermediate velocity is calculated without considering the pressure gradient term in the URANS equation. Next, with the intermediate velocity field, the pressure is calculated from the Poisson equation using the Jacobi-preconditioned BiCGStab solver [13]. Finally, the pressure term is used to update the velocity field. In addition, the implementation of staggered grid method ensures a tight coupling between pressure and velocity fields.

The free surface computations are performed with the level set method. It is defined as the signed distance function $\phi(\vec{x}, t)$ [14] and it provides the distance to the interface and also the sign as follows:

$$
\phi(\vec{x}, t) =
\begin{cases}
> 0 & \text{if } \vec{x} \in \text{water} \\
0 & \text{if } \vec{x} \in \Gamma \\
< 0 & \text{if } \vec{x} \in \text{air}
\end{cases}
$$

(3)

A convection equation for the level set function is:

$$
\frac{\partial \phi}{\partial t} + u_j \frac{\partial \phi}{\partial x_j} = 0
$$

(4)

The Eikonal equation $|\nabla \phi| = 1$ is valid in the computational domain. The URANS equations are closed with the two-equation $k-\omega$ model [15]. A three-dimensional ghost cell immersed boundary method (GCIBM) [16] is implemented to model the complex geometries. Parallel computation based on the domain decomposition method and MPI (Message passing interface) is implemented in the numerical model.

2.1 Wave generation

In the laboratory experiments performed by Svangsttu (2011) [17], a flap-type wave generator was used to generate focused waves. In order to mimic the experimental condition, a single hinged flap-type wave maker theory is implemented in the numeral model. In the numerical
wave tank, focused waves are generated using the recorded time history of paddle motion of the physical wave flume. A flap wave maker is hinged at a distance of \( z_s = 0.06 \text{m} \) from the bottom with a height of \( z_e = 1.36 \text{m} \) and it corresponds to rotating angle \( \theta \) as shown in Fig. 1. At the undisturbed free surface \( (z=0) \), the flap horizontal displacement is \( x_f \) and the velocity is \( u_f \). The horizontal displacement \( (x_f) \) of a flap decreases linearly towards the hinge point at which the displacement is zero.

\[
x_f = \left( z - z_s \right) \frac{a_f}{z_e - z_s} \sin \theta
\]

The angle \( \theta \) and \( a_f \) are related as follows:

\[
tan \theta = \frac{a_f}{z_e - z_s}
\]

At the outlet boundary, the active beach [18] is implemented to absorb waves. REEF3D [8] has been used to study a wide range of coastal and offshore problems. The numerical model is well utilized for modelling breaking waves in shallow waters [19, 20], non-breaking and breaking wave interaction with vertical cylinders[21, 22, 23, 24].

3 RESULTS AND DISCUSSION

3.1 Numerical set-up and grid refinement study

The numerical set-up consists of a 1.5m long flatbed followed by a slope of 1/20 with a water depth of 1.0m. as shown in Fig. 2. The coordinate system is the same as the laboratory experiments. The wave maker consists of a paddle of 1.36 m long that is hinged (at \( z=0.06 \text{m} \)) on a raised support structure. At the wave inlet, the measured time history of paddle displacement is given as input to generate focused waves. In order to examine the effect of grid size on the numerical results, three grid sizes, \( dx=0.01 \text{m}, 0.025 \text{m} \) and 0.05m are tested in the present study. Fig. 3 show the comparison of experimental and numerical wave surface elevations at \( x=4.75 \text{m} \) (WG1) and 12.21m (WG3) for different grid sizes \( dx=0.01 \text{m}, 0.025 \text{m} \) and 0.05m. The computed focused wave crest with \( dx 0.05 \text{m} \) at both wave gauges are lower and the phases are also different than the experimental data 3. The numerical results with \( dx=0.01 \text{m} \) and 0.025m show good agreement with the measured data at both wave gauges. Though with \( dx=0.01 \text{m}, \)
experimental waves are accurately represented including the asymmetric focused wave profile before breaking at $x=12.21$ m (WG3). Therefore the grid size $dx=0.01$ m is considered for the present study.

3.2 Breaking focused wave characteristics

Focused waves are generated using a single hinged flap maker theory as discussed in Sec. 2.1. Fig. 4 shows comparison of numerical results with the experimental data for the wave surface elevations along the wave tank at $x=4.75$ m (a, WG1), 11.66 m (b, WG2), 12.51 m (c, WG3) and 12.95 m (WG4). It appears that the computed wave surface elevations are in good agreement with the measured results for all wave gauges. When the focused wave group propagates over the slope in decreasing water depth, the focused as well as the secondary wave crests transform into narrower and sharper due to shoaling which causes the shallow water steepening and shortening of waves. At $x=4.75$ m, the focused wave group are started to shoal without much deformations. During the initial stages of shoaling, potential energy slightly increases as the wave height increases and thus, kinetic energy slightly decreases.

As the wave group propagates further over the slope, the speed of the wave group is retarded and the wave height further increases at $x=11.66$ m (WG2). The wave group consists of a number of waves of different frequencies are started superimposing upon each other causing a single large focused wave crest as shown in Fig. 4 (b) $x=11.66$ m (WG2) and (c) 12.51 (WG3). When the focused wave crest reaches the maximum height, particle velocity in the upper part of the wave crest increases and thus, kinetic energy. Finally, the steep wave front becomes nearly vertical and eventually the main focused wave crest breaks. Fig. 4 (d) shows the free surface elevation of a nearly breaking wave crest at $x=12.51$ m (WG3). In the laboratory experiments, the focused wave crest breaks between $x12.51$ m (WG3) and 12.91 m (WG4) as the exact breaking location is not available. Whereas in the numerical simulation, the wave breaks exactly at $x_b=12.62$ m. The computed breaker water depth ($d_b$) and the breaker height are 0.46 m and 0.22 m, respectively. As expected, the computed focused wave height decreases after breaking as presented in Fig. 4 (e) due to energy loss during breaking. Moreover, the computed free surface elevations during shoaling, breaking and post-breaking stages are in good agreement with the experimental data. Fig. 5 shows comparison of numerical free surface profiles with the experimental data for the
Figure 4: Computed wave surface elevations along the wave tank at $x=4.75\text{m (WG1)}$, $11.66\text{m (WG2)}$, $12.21\text{m (WG3)}$, $12.51\text{m (WG4)}$ and $12.95\text{m (WG5)}$; circles: experimental data [17] and red solid lines: numerical results.
evolution of the focused wave group along the wave tank. The wave components start to focus as the group propagates over the slope and the amplitude of the focused wave crest increases. The focus wave height increases continuously up to the breaking point and the focused wave crest breaks at $x_b=12.62$m. After the breaking point, the wave heights suddenly decreases shoreward. The propagation of the wave group and the development of the focused wave crest during shoaling and breaking are consistent with the experimental data [17].

3.3 Free surface flow features of breaking focused waves

Fig. 6 presents the simulated free surface profiles with velocity magnitude (m/s) at $t=41.45$s, 41.80$s, 41.90$s and 42.0s after breaking. As mentioned in Sec 3.2, the focused wave crest evolved from the wave group breaks at $x=12.62$m with a large wave front. At the incipient of breaking, the front face of the focused wave crest becomes vertical. As the wave group propagates further over the slope the upper part of the focused wave crest with high velocity travels faster than the rest of the wave. The wave front overturns and ejects forward in the wave direction as shown in Fig. 6 (a) and (b). Then the overturned wave crest falls down into the forward wave trough and causes a enclosed air-pocket as shown in Fig. 6 (c). Further the translation of the overturned crest displaces a portion of water shoreward causing the splash-up and the formation of secondary waves crest as depicted in Fig. 6 (d). It should be noted that the velocity magnitude of the secondary wave crest is almost equal to the velocity of the main wave crest. The simulated flow features such as the development of overturning wave crest and its impingement and the formation of air-pocket, the splash-up and the secondary wave crest are well captured in the present numerical simulations. Moreover, the numerically captured free surface flow features are consistent with the previous numerical studies [?, 20] and experimental studies [?, ?] on breaking waves over slopes.

4 CONCLUSIONS

Breaking focused waves are simulated in the numerical wave tank based on the CFD model REEF3D. The numerical model describes the two-phase flow using the incompressible Reynolds-Averaged Navier-Stokes (RANS) equations together with the continuity equation. The free
surface is modeled with the level set method. Turbulence is described with the two-equation \( k - \omega \) model. The computed results are compared with the experimental data for wave surface elevations over the slope and the development of the free surface profiles along the wave tank during the breaking process. The experimental and theoretical comparison showed to be in a good agreement. Further, the transformation of the focused wave group over a slope is also presented and discussed. The simulated free surface flow features are consistent with the previous studies on regular breaking waves over slopes. The present numerical study provides some insight into the modeling aspects of breaking focused waves and the physical processes related to the evolution of breaking focused waves over a sloping seabed. The present work has investigated some aspects concerning the characteristics of breaking focused waves. However, more research is needed to understand the complete physical processes involved during the interaction of breaking focused waves with structures and the related flow characteristics including the wave impact forces.

5 ACKNOWLEDGMENT

The research work has been funded by the Research Council of Norway through the project ”Hydrodynamic Loads on Offshore Wind Turbine Substructures due to Nonlinear Irregular Breaking, High Steep and Extreme Waves” (project number: 246810). The authors gratefully acknowledge the computing time granted by NOTUR (project number: NN2620).

REFERENCES


Figure 6: Free surface flow features with velocity magnitude after breaking for different time instants at $t = 41.45s$ (a), 41.80s (b), 41.90s (c) and 42.0s (d).
SIMULATION OF FISH ESCAPE AND SWIMMING TOWARD A PREDEFINED GOAL

Seyed-Amin Ghaffari*, Stéphane Viazzo††, Kai Schneider† and Patrick Bontoux††

* Société Hydro-Mécanique Marine, 13127 Vitrolles, France
e-mail: s.amin.ghaffary@gmail.com

† I2M - UMR 7373 - CNRS, Aix-Marseille Université
Centre de Mathematiques et d’Informatique
39 rue Joliot-Curie, 13453 Marseille, France
e-mail: kai.schneider@univ-amu.fr

†† M2P2 - UMR 7340 - CNRS, Aix-Marseille Université
38 rue Joliot-Curie, 13451 Marseille, France
Web page: http://www.m2p2.fr

Key words: Fluid interaction with forced deformable bodies, Compact finite difference, Volume penalization, Fish escape, Wake of a swimming fish

Abstract. We present an immersed boundary method for numerical simulation of a swimming fish. The vorticity transport equation is solved on a Cartesian grid using compact finite differences. A Lagrangian structured grid defines the fish body and is moving in the surrounding incompressible flow due to the exerted hydrodynamic forces and the torque. An efficient law determining the curvature of a swimming fish is presented which is based on the geometrically exact theory of nonlinear beams and quaternions. Validation of the solver shows the efficiency and expected accuracy of the algorithm for swimming fish simulations. The structure of the wake of a swimming fish is studied, and some common features with the wake of a flapping plate are demonstrated.

1 INTRODUCTION

Investigation about swimming can help researchers to provide insights into nature, biology, fluid mechanics, fluid structure interaction, etc. On the other hand, this can help engineers to design fast boats, innovative bioinspired propulsion systems, like robofishes and kayaks fitted with new propulsion systems. As an example a Mirage Drive paddle system is used in a new kayak, in which two underwater flexible flippers moving to right and left, producing a forward propulsion. In this regard, many experimental and numerical studies were performed considering the hydrodynamics of swimming.
Three-dimensional numerical simulation of swimming have only become possible in the last decade. Even if due to the shape and deformation style of the fish-like swimmers the surrounding flow is fully three-dimensional, most of the fundamental features of swimming are included in two-dimensional analyses. Therefore, in the present investigation we study the rotation and escape of a two-dimensional swimmer. The well accepted mathematical model for incompressible flows are the Navier–Stokes equations. The presence of nonlinearity in the governing equations restricts the analytical solutions to simple domains. In recourse to numerical simulations of the Navier–Stokes equations, according to many researchers the vorticity stream-function formulation proved to be efficient for two-dimensional simulations.

A common discussion in numerical simulations would be the grid spacing in accordance with the Reynolds number. For high Reynolds number flows a direct numerical simulation (DNS), i.e., using a grid spacing capable to capture all length scales present in the flow field, would not be a possible practice. Moreover, the nonlinearity promotes the creation of small scales. The small scales (subgrid scales) need to be modeled to avoid blow-up (instabilities) in the flow field. A common practice is to use a turbulence model via adding an eddy viscosity. Upwinding, adding artificial dissipation or applying a low-pass filter to the flow field are equivalent tricks. In the present methodology, an approach similar to implicit LES is applied to the flow field, to dissipate the transferred energy toward high wave numbers. We apply a low-pass spatial-filtering in accordance with the high-order compact differencing to the equations. This enables us to use a tailored grid when high Reynolds flow simulations are envisaged.

Simulation of swimming lead to a fluid-structure interaction problem, i.e., from numerical view point we have a moving boundary problem. By using an immersed boundary method (IBM), see e.g. Mittal and Iaccarino [2] and Schneider [7], a Cartesian grid and therefore a high-order and efficient method can be used. This considerably increases the accuracy far from the solid boundaries, on the other hand the accuracy near moving boundaries remains acceptable, i.e., between first and second order accuracy can be achieved. However, some researchers developed higher-order IBMs, see for example Linnick and Fasel [3]. In the present investigation, the motion of the fish is imposed, thus we reconstruct the boundaries of the fish at each time step. We then use Newton–Euler equations to move and rotate the body of the fish around the reference point. The boundary of the fish is described exactly within the grid space accuracy.

In solving the incompressible Navier–Stokes equations an elliptic Poisson equation is typically encountered which is the most time consuming part of the algorithm. In the vorticity stream-function formulation, an elliptic equation has to be solved with Dirichlet boundary condition for vorticity and stream-function. We are using the fourth-order direct method proposed by Ghaffari et al. in [6] for an efficient solution of the Poisson equation.

2 METHODOLOGY

The governing equations of incompressible flows are the Navier–Stokes equations. For two-dimensional flows, by taking the curl of the Navier–Stokes equations, the vorticity transport equation can be derived as follows:

$$\partial_t \omega + (\mathbf{u} \cdot \nabla) \omega = \nu \nabla^2 \omega + \nabla \times \mathbf{F} , \quad \mathbf{x} \in \Omega \in \mathbb{R}^2$$

where \( \omega(\mathbf{x}, t) = \nabla \times \mathbf{u} = v_x - u_y \) denotes the vorticity, \( \Omega \) is the spatial domain of interest, \( \mathbf{u}(\mathbf{x}, t) \) is the velocity field, \( \nu = \mu / \rho_f > 0 \) is the kinematic viscosity of the fluid, \( \rho_f \) is the density.
and $\mathbf{F}(\mathbf{x}, t)$ is a force function. For a complete description of a particular problem, the above equations need to be complemented to describe an initial/boundary value problem. The velocity components are $(u, v) = (\partial_y \psi, -\partial_x \psi)$, with $\psi$ satisfying the Poisson equation

$$-\nabla^2 \psi = \omega$$

(2)

To take into account a deformable body which moves in the solution domain the volume penalization method is used. The volume penalization method belongs to the family of immersed boundary methods, we refer to Mittal and Iaccarino [2] and Schneider [7] for more details. The forcing function is representative of the immersed body and is defined as follows:

$$\mathbf{F} = -\eta^{-1} \chi (\mathbf{u} - \mathbf{u}_B)$$

(3)

where $\mathbf{u}_B(\mathbf{x}, t)$ is the velocity field of the immersed body. The penalization parameter $\eta$ represents the permeability coefficient of the immersed body. The mask (or characteristic) function $\chi$ is dimensionless and describes the geometry of the immersed body

$$\chi(\mathbf{x}, t) = \begin{cases} 
1 & \mathbf{x} \in \Omega_B \\
0 & \mathbf{x} \in \Omega_f 
\end{cases}$$

(4)

where $\Omega_f$ represents the domain of the fluid and $\Omega_B$ represents the immersed body in the domain of the solution (see Fig. 1). The solution domain $\Omega = \Omega_f \cup \Omega_B$ is governed by the Navier-Stokes equations in the fluid regions and by a Darcy–Brinkmann law in the penalized regions, in the limit when $\eta \to 0$. For fluid/solid interaction problems the simulations start with the body $\mathbf{u}_B(\mathbf{x}, 0) = 0$, and fluid at rest, i.e., $\omega(\mathbf{x}, 0) = \psi(\mathbf{x}, 0) = 0$ and free-slip boundary conditions are imposed at the surrounding walls ($\psi|_{\partial \Omega} = \omega|_{\partial \Omega} = \partial p/\partial \mathbf{n}|_{\partial \Omega} = 0$). For time integration we are using classical fourth-order Runge–Kutta method. The spatial discretization and filtering are based on compact finite differences, we refer to Ghaffari et al. [6] for more details.

3 APPLICATION AND RESULTS

To perform simulations of a fish swimming toward a predefined goal, a domain of $(x, y) \in [0, 5l_{\text{fish}}] \times [0, 5l_{\text{fish}}]$ is chosen, with $l_{\text{fish}} = 1$. The Reynolds number is defined as $Re = \overline{U} l_{\text{fish}} / \nu$, where $\overline{U}$ is the mean bulk velocity of the fluid.
where \( \bar{U} \) is the average swimming speed and \( \nu \) is the kinematic viscosity of the fluid. The backbone of the fish is performing an imposed undulatory periodic motion according to

\[
y(x, t) = (a_0 + a_1 x + a_2 x^2) \sin\left(2\pi \left(\frac{x}{\lambda} + f t\right)\right)
\]

where \( \lambda = f = 1, a_2 = 0, a_1 = 0.1212, a_0 = 0.0037878 \). Considering the fact that curvature is a more convenient framework for geometry construction, from two successive derivatives of Eq. (5) the needed curvature for the propulsion mode can be derived. We refer to Boyer et al. [4] for details of the backbone definition with the use of quaternions. The motion of the backbone is starting smoothly with a cosine type rise function \( x \in [\pi, 2\pi] \), in the first period and tends to zero in the same manner after reaching the goal. The geometry of the fish is defined by the half width of the body along its arclength, which is superimposed on the curve of the backbone given by Eq. (5). We refer to Carling et al. [1] for details of geometry definition and to Ghaffari et al. [6] for details on rotation control. An important point in swimming is to identify the origins and evolution of the vortices. A street of single and pair vortices constructs the morphology of the wake. The vortices are stronger near the swimming fish and are weaker away from the fish because of the damping effect of fluid viscosity. The motion of the vortices is a self induced movement. They repulse or attract each other. They can merge according to their sign and surrounding vortices. The weaker ones are more subjected to merge with the stronger ones. Depending on the Reynolds and Strouhal numbers, the tail-beat amplitude and the trajectory of the fish, different wakes can be observed. According to Schnipper et al. [5] from 2 up to 16 vortices per oscillation period can be created in pair or single configurations. As a test case, a simulation of a fish swimming toward a predefined goal, located exactly in its backside, is considered (\( Re \approx 3500 \)). In our simulations, in each stroke (one tail-beat) approximately 4 vortices are created. We observe two strong and two weak vortices. Within 15 strokes (\( t = 15 \)), 10 pairs (dipole) and 10 single vortices can approximately be identified (see Fig. 2). In the second test case (\( Re \approx 5000 \)) the fish is swimming toward a predefined goal located in the front. Within 20 strokes (\( t = 20 \)), 12 pairs (dipole) and 15 single vortices can approximately be identified (see Fig. 3). Some weaker vortices are merging with the stronger ones just after their creation in the wake. The pair (dipole) vortices are stronger than the single ones, thus persisting longer in the flow field. The wake is stable during the swimming and becomes unstable after reaching the goal and stopping the stroke.

4 CONCLUSIONS

- The structure of the wake of a swimming fish is studied, and some common features with the wake of a flapping plate are demonstrated. A street of single and pair vortices constructs the morphology of the wake.

- In each stroke (one tail-beat) approximately 4 vortices are created. We observe two strong and two weak vortices.

- Some weaker vortices are merging with the stronger ones just after their creation in the wake.

- The pair (dipole) vortices are stronger than the single ones and thus persist in the wake even after stopping the stroke.
Figure 2: Vorticity fields of a fish swimming toward a predefined goal, located in its backside, $Re \approx 3500$. Within 15 strokes ($t = 15$), 10 pairs (dipole) and 10 single vortices can approximately be identified.

- The wake is stable during the swimming and becomes unstable after reaching the goal and stopping the stroke.

REFERENCES


Figure 3: Vorticity fields of a fish swimming toward a predefined goal, located in the front, $Re \approx 5000$. Within 20 strokes ($t = 20$), 12 pairs (dipole) and 15 single vortices can approximately be identified.
A 3D RANS-VOF WAVE TANK FOR OSCILLATING WATER
COLUMN DEVICE STUDIES

MARINE 2017

PAULO R.F. TEIXEIRA*, ERIC DIDIER†+ AND MARIA G. NEVES†&

* Universidade Federal do Rio Grande (FURG)
Engineering School, Av. Itália, km 8
Campus Carreiros 96201-900 Rio Grande, RS, Brazil
e-mail: pauloteixeira@furg.br, www.furg.br

† National Laboratory for Civil Engineering (LNEC)
Harbours and Maritime Structures Division
Av. do Brasil 101, 1700-066 Lisbon, Portugal
email: edidier@lnec.pt; gneves@lnec.pt, www.lnec.pt

+ Universidade Nova de Lisboa, Faculdade de Ciência e Tecnologia (FCT-UNL)
UNIDEMI
Campus de Caparica, 2829-516, Monte de Caparica, Portugal
e-mail: edidier@lnec.pt, www.fct.unl.pt

& Universidade Nova de Lisboa, Faculdade de Ciência e Tecnologia (FCT-UNL)
Civil Engineering Department
Campus de Caparica, 2829-516, Monte de Caparica, Portugal
email: gneves@lnec.pt, www.fct.unl.pt

Key words: Oscillating Water Column, Numerical Modelling, RANS-VOF, Wave Dynamic Absorption

Abstract. This paper presents the validation of a wave maker with active absorption method implemented in FLUENT® RANS-VOF code for modelling long time series of waves propagating and interacting with coastal structures, more specifically, with Oscillating Water Column devices, to cancel out the reflected wave that reaches the wave maker. Verification of the numerical technique was performed in a 3D wave tank in which a technique of multiple active absorption wave makers was used. Good results were obtained in quasi-3D and fully-3D applications showing that active absorption should be used for modelling accurately the complex interaction between wave and coastal structures, in the present case, with an oscillating water column device integrated in a vertical breakwater.

1 INTRODUCTION

The performance of an oscillating water column (OWC) wave energy converter depends on many factors, such as the incident wave condition, the tidal level and the coupling between the chamber and the air turbine. Other factors are related to the location of the device (offshore, nearshore or onshore): effect of the water depth, diffraction and reflection effects,
on the OWC and the nearest structures (breakwater, coast, etc.).

Several 2D and 3D numerical studies of the performance of offshore and onshore OWCs have been carried out by using numerical models with different complexity, such as the potential flow models, in which the Boundary Element Method (BEM) is commonly applied [1, 2, 3], and the Reynolds-Averaged Navier–Stokes (RANS) models, in which Finite Element Model or Finite Volume Model are employed with appropriate methods to capture the movements of free surface [4, 5, 6, 7]. The main difficulty of these numerical investigations is to accurately impose the wave generation, the wave dissipation at the end of wave flume/tank (to avoid wave reflection) and the dynamic wave absorption at the wave generation boundary (to cancel out the reflected wave that reaches the wave maker leading to re-reflections at the wave maker).

The present paper describes the development of a 3D numerical wave tank using RANS–VOF FLUENT® numerical model [8], which is based on the RANS equations and the Volume of Fluid (VOF) surface capturing method. Specifically, a methodology which takes into account the dynamic absorption at wave generation is implemented in FLUENT®. Active wave absorption is widely known in physical modelling, as it is needed to cancel out reflected waves that reach the wave maker. The methodology proposed by Shäffer and Klopmann [9] is followed since it is the easiest technique to implement, based on linear shallow water theory, and has shown that it works relatively well even when used for waves outside the shallow water range. This technique was implemented in previous works on other numerical models, such as IH-2VOF by Lara et al. [10] and OpenFOAM® by Higuera et al. [11], RANS type models using a VOF technique for free surface flow, and in SPHysicsCE (Smoothed Particle Hydrodynamics) model by Didier and Neves [12]. The active wave absorption enables also using a small numerical tank (low computational cost) and a long data capture for accurate analysis of statistical parameters.

The first application presented in this paper consists in modelling a quasi-3D wave tank corresponding to the particular configuration for which the width of the wave tank is equal to the width of the OWC chamber, i.e. the computational domain is reduced to a 2D wave flume. Verification of the active absorption method at the wave maker is carried out for an OWC device at the end of the flume, with and without active absorption.

The second application considers a fully-3D wave tank for modelling the interaction between an incident wave and an OWC device inserted in a vertical breakwater. Multiple active absorption wave makers are used since the flow is fully 3D. Moreover, for checking its efficiency, results are compared with results of a single active absorption wave maker.

2 SET OF EQUATIONS

The incompressible fluid flow can be described by the continuity and Navier-Stokes equations:

\[
\frac{\partial u_i}{\partial x_i} = 0
\]

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + g_i + \frac{1}{\rho} \frac{\partial \tau_{ij}}{\partial x_j}
\]
where \( i, j = 1, 2 \) for 2D flow and \( i, j = 1, 2, 3 \) for 3D flow, \( u_i \) refers to the velocity components, \( t \) the time, \( p \) the pressure, \( \rho \) the fluid density, \( g_i \) the gravitational acceleration components, and \( \tau_{ij} \) the viscous stress tensor. In this work, Reynolds-Average Navier-Stokes (RANS) equations, based on the decomposition of the instantaneous velocity and pressure fields of the Navier–Stokes equations (Eqs. 1 and 2) into mean and fluctuating components, and the subsequent time-averaging of the set of equations, are used. This process introduces Reynolds stresses terms associated with the turbulence. A turbulence model must be introduced to relate the Reynolds stresses to mean flow variables and close the equations. The standard \( k-\varepsilon \) turbulence model was used. Free surface flow motion was defined by the Volume of Fluid (VOF) method [13] which consists, through the transport equation of a scalar, the volume fraction \( \alpha \), (Eq. 3) to identify the position of the free surface from the volume fraction which takes the value 0 in the air and 1 in the water. The position of the free surface is defined by the value 0.5.

\[
\frac{\partial \alpha}{\partial t} + u_j \frac{\partial \alpha}{\partial x_j} = 0
\]

3 NUMERICAL MODEL

The 2D/3D RANS-VOF numerical code FLUENT®, version 6.3.26 [8] applies a Finite Volume technique to solve the RANS and the VOF equations.

Integration in time is performed using the implicit formulation and the 2nd order time discretization. The diffusive terms of the equations are discretized by the second order central difference scheme. The convective terms in the faces of the control volumes, for the components of momentum, are determined by the MUSCL scheme, while second order upwind scheme is used for convective terms of \( k \) and \( \varepsilon \) equations. The pressure is determined by the PRESTO! (PREssure STaggering Option) scheme, classically used for wave propagation modeling in FLUENT® [8]. The standard \( k-\varepsilon \) turbulence model, with default coefficients, is choose since it allows an accurate wave propagation and takes into account the principal phenomenon of turbulent flow. The SIMPLEC algorithm is used for coupling pressure and velocity and under-relaxation coefficients are equal to 1 for momentum and VOF and 0.8 for \( k-\varepsilon \). In the VOF method, both water and air are considered incompressible and the volume fraction is defined by a modified version of the High Resolution Interface Capturing (HRIC) scheme [14].

To model wave interaction with an OWC device, the following boundary conditions are imposed: atmospheric pressure at the top of the wave flume; static wave maker at the inlet; wall and no-slip condition at the coastal structure; no-slip condition at the bottom. Initial conditions are hydrostatic pressure in the water, still water level and flow at rest (zero velocity). Time step is equal to \( T/600 \) (where \( T \) is the wave period) and six non-linear iterations per time step enable to reduce the residue by at least two orders of magnitude which is enough for good accuracy [4, 5, 15].

Previous studies have shown that wave propagation in FLUENT® is well simulated using 60-70 elements per wavelength in the horizontal direction, and 20-25 elements in the vertical direction in the region of the free surface [4, 5, 15]. Consequently, the Cartesian mesh is constructed with these characteristics with a refinement near the static wave maker, the wall
of the coastal structures and the flume bottom. Figure 1 shows a typical mesh used in the 2D flume (Fig 1a), with refinement around the still water level \((x_2 = 0 \text{ m})\) and the wave maker \((x_1 = 0 \text{ m})\), and in the 3D tank (Fig. 1b), with refinement at the wall of vertical breakwater and OWC device and still water level (mesh in the horizontal plane, \(x_3 = 0 \text{ m}\)).

![Typical mesh near a OWC device for 2D - wave flume (a) and 3D - wave tank (b)](image)

### 3.1 Wave generation

Wave generation was performed using a static wave maker at the computational domain. Velocity component profiles, which are in function of time and depth, are imposed and the corresponding free surface position is defined by the volume fraction value, 0 above and 1 under the free surface. In FLUENT®, velocity and volume fraction profiles at inlet are implemented through the DEFINE_PROFILE UDF (User Defined Function).

### 3.2 Wave generation with active absorption

Active wave absorption is widely known in physical modelling, as it is needed to cancel out reflected waves that reach to the wave maker. The methodology proposed by Shäffer and Klopman [9] is followed. It is an easy technique to implement, based on linear shallow water theory [16] and consists correcting the velocity at the wave maker by defining the reflected wave height. This procedure was implemented successfully in previous works on other numerical models, such as IH-2VOF [10] and OpenFOAM® [11], RANS-VOF models, and SPHyCE (Smoothed Particle Hydrodynamics) model [12], and have shown that it works relatively well even when is used for wave conditions outside the shallow water range.

In this methodology, the horizontal velocity component at the wave maker is corrected in real time to avoid the wave reflection at the wave maker. For that, it is necessary to estimate the free surface elevation of the reflected wave, \(\eta_R\), to be absorbed, comparing the target-free surface, \(\eta_{\text{target}}\), calculated using linear wave theory, to the free surface recorded in front of the wave maker, \(\eta_{\text{record}}\):

\[
\eta_R = \eta_{\text{target}} - \eta_{\text{record}}
\]  

(4)

The velocity component \(u_1\) (on the wave propagation direction) at the wave maker has to be modified in order to match the velocity induced by the wave to be absorbed. The velocity correction owing to absorb the reflected wave, \(u_{\text{corr}}\), can be written as follows:

\[
u_{\text{corr}} = \eta_R \left(\frac{g}{d}\right)^{1/2}
\]  

(5)

where \(d\) is the water depth. The corrected velocity at the wave maker, \(u_1\), is obtained by:
\( u_1 = u_{\text{target}} - u_{\text{corr}} \) (6)

The free surface elevation in front of the static wave maker and the velocity correction are defined using the DEFINE _ADJUST UDF and velocity components and volume fraction profiles at the inlet are imposed using the DEFINE_PROFILE UDF. Active wave absorption is applied on a single wave maker or an outgoing wave boundary for 2D/3D applications in wave flumes, and on multiple wave makers or multiple paddles, acting at each paddle independently, for applications to fully-3D wave tank [17, 18].

However, absorption for fully-3D wave tank needs to take into account the additional information of wave direction in front of each paddle. The corrected velocity, \( u_{\text{corr}} \), due to the direction of incident wave, involves the calculation of mean horizontal velocity, \( u_{\text{corr}} \) (Eq. 5), and its mean direction. The velocity is decomposed in a component normal to the paddle, i.e. the corrected velocity, \( u_{\text{corr}} \), and another tangential to it, \( u_{\text{tg}} \):

\[ u_{\text{corr}}^2 = u_{\text{corr}}^2 + u_{\text{tg}}^2 \] (7)

Therefore, the corrected velocity that can be only applied on the perpendicular direction to the paddle [11], is obviously defined by:

\[ u_{\text{corr}} = (u_{\text{corr}}^2 - u_{\text{tg}}^2)^{1/2} \] (8)

It can be noticed that, when the tangential component of the velocity is greater than the total horizontal velocity, solution of Eq. 8 is not real. So that, if the value inside the square root is negative, \( u_{\text{corr}} \) is set to zero and absorption is not applied. The corrected velocity at the wave maker, \( u_1 \), is obtained from the Eq. 6 by:

\[ u_1 = u_{\text{target}} - u_{\text{corr}} \] (9)

The active absorption method was validated for 2D and 3D applications for wave flume and wave tank for vertical breakwaters [17, 18].

3.3 Wells turbine model

The Wells turbine at the top of the OWC device chamber is modelled by imposing the pressure, \( p_t \), in function of the flow rate, \( Q_t \), calculated at the same boundary, according to Eq. 10, where \( k_t \) is the turbine characteristic and \( p_o \) the reference atmospheric pressure.

\[ p_t - p_o = k_t Q_t \] (10)

The flow rate needs special attention, since, for 3D applications, it is obviously directly calculated at the boundary of the OWC air chamber. However, for quasi-3D applications, i.e. 2D computational domain considering that the width of the tank is equal to the width of the OWC chamber, the aerodynamic effect of the Wells turbine in the air chamber, particularly the pressure and its action on the free surface inside the OWC chamber, is 3D and, consequently, the flow rate at the turbine is calculated taking into account the width of the chamber [19].

4 RESULTS AND DISCUSSION

The presented wave generation with active absorption of reflected waves method is validated for quasi-3D and fully-3D applications for an incident regular wave interacting with an OWC located at the end of the wave tank.
4.1 Quasi-3D wave tank

The simulation is carried out for a quasi-3D wave tank corresponding to the particular configuration for which the width of the wave tank is equal to the width of the OWC chamber, i.e. the computational domain is reduced to a 2D wave flume. Verification of the active absorption method at the wave maker is carried out for an OWC device at the end of the flume. The OWC is represented by a 10 m length, 10 m wide chamber. The local water depth, \( d \), is 10 m, the front wall submergence depth (lip) is set to 5 m and its thickness 0.5 m. Incident regular wave has a wave period of \( T = 9 \) s and a wave height of \( H = 1.0 \) m which corresponds to wave propagation in intermediate water depth. The length of the flume is set to 81.8 m that corresponds to the wave length, \( L \), for this water depth.

Efficiency of active absorption is demonstrated comparing a simulation with and without active absorption. Time series of free surface elevation are analyzed at two gauges: G1, located 1.0 m from the wave maker and G2, located at 1.0 m from the OWC wall. The mean free surface elevation inside the water OWC chamber, \( \eta_{OWC} \), referred as \( G_{OWC} \), is also analyzed. The pneumatic power, \( P_p \), is calculated from the flow rate at the turbine and the mean pressure inside the OWC air chamber.

Figure 2 shows the time series of free surface elevation at gauge G1 and G2, mean free surface elevation inside the water chamber and pneumatic power with and without active absorption. Time series of free surface elevation at G1 and G2, Figs. 2(a) and (b), respectively, show clearly the necessity of using active absorption at wave maker to obtain accurate results. It can be noted that, using active absorption, free surface elevation reaches quickly a regular periodic motion at G1 and G2 with regular amplitude of crest and through, near the wave maker and in front of the OWC wall, as expected. This is not the case without absorption. Inside the OWC, same comments can be made for the mean free surface elevation.

![Figure 2: Time series of free surface elevation at gauges G1 (a), G2 (b) and G_{OWC} (c) and pneumatic power (d) with and without active absorption](image-url)
Due to the inaccurate mean free surface elevation inside the OWC chamber without active absorption, pressure inside the air OWC chamber and flow rate at the turbine boundary are not correctly defined for the true incident wave characteristic such as the calculated pneumatic power. Consequently, mean pneumatic power using active absorption at wave maker is 67 kW and without active absorption it is overestimated, 86 kW.

4.2 Fully-3D wave tank

Studies of OWC devices are classically carried out in a wave flume, where the width of the OWC chamber being equal to the width of the flume. However, onshore OWC devices can be integrated in vertical breakwaters, such as Sakata in Japan [20] and Mutriku in Spain [21], or at the coastline, like Pico in Portugal [22] and Limpet in Scotland [23]. A fully 3D study of this type of OWC devices needs taking into account the coastline and/or the coastal structures at the proximity of the device and, consequently, requires using a fully-3D wave tank.

In the present study, the wave tank, Fig. 3, is 98.1 m long, 35 m wide and 10 m deep. The vertical breakwater is 32.5 m long. The OWC is represented by a 5 m length, 5 m wide chamber and the front wall submergence depth (lip) is set to 5 m and its thickness is 0.5 m. Symmetry condition is imposed on the left and right sides of the wave tank and on the OWC, Fig. 3. With these conditions, an array of OWC devices (65.0 m spaced, i.e. ~L) integrated in a vertical breakwater is modeled. Static multiple active absorption wave maker is used, with 7 paddles with 5 m width each. However, for checking its efficiency, simulation is also performed using a single active absorption wave maker. Wall condition is applied on the wave tank bottom, OWC walls and vertical breakwater. In this application, the OWC air chamber is fully open, i.e. without turbine. Position of free surface elevation gauges at 1 m from the wave maker and 1 m in front of OWC and vertical breakwater is also indicated in Figure 3.

Incident regular wave has a period of $T=7.5$ s, a wave height of $H=1.0$ m, which corresponds to wave propagating in intermediate water depth, and to a wave length of $L=65.4$ m. Time step is equal to 0.0125 s and six non-linear iterations per time step are performed. The mesh is composed by 336462 hexahedral control volumes. CPU time in serial is about 205 min per wave period on PC Intel(R) Core(TM) i7-3820 CPU @ 3.60 GHz.

Figure 4 shows the free surface elevation at the wave tank at $t=250.0$ s, for single and multiple active absorption wave makers, respectively. Efficiency of multiple active absorption wave makers is obvious since the regular incident wave is well conserved along the time, showing a crest and trough nicely parallel to the wave maker. For a single wave maker, an oscillation appears in the tank, due to the generation of “false waves” produced by the single
active absorption wave maker itself that does not correctly cancel out the reflected waves that come from the OWC and vertical breakwater.

Figures 5 and 6 show the free surface elevation along the wave maker, at gauges G01 to G04, and along the vertical breakwater, at gauges G12 to G14, for a single and a multiple active absorption wave makers, respectively. It can be seen, from Figs. 5 and 6, the effective efficiency of the multiple active absorption wave makers when compared to the single one. In Fig. 5, free surface elevation at the wave gauges in front of the wave maker shows a variability of wave height, from 0.7 to 1.4 m, indicating the presence of re-reflected waves in the tank. The single active absorption wave maker generates a ‘false’ incident wave due to the deficient canceling of true reflected waves at the wave maker. Consequently, free surface elevation and, consequently, wave heights along the vertical breakwater present a large variability, with wave heights at gauges G12 to G14 being very different from the predictable wave height of 2.0 m, corresponding to typical standing waves due to the presence of a vertical wall.

With multiple active absorption wave makers, Fig. 6, free surface elevation at the wave maker is relatively regular and the wave height is around 2.0 m, due to the reflection of vertical breakwater and OWC, as expected. It can also be observed that the free surface elevation at the breakwater is regular at each of the gauges and the wave height is around 2.0 m, typical of fully reflected wall and standing waves.

Fig. 7 shows the time series of mean free surface elevation inside the OWC water chamber, \( \eta_{\text{OWC}} \), and the free surface elevation in front of and outside the OWC wall, at gauge G11, for a single and multiple active absorption wave makers. It can be seen, Fig. 7b, that the wave height in front of the OWC wall, using a multiple active absorption wave makers, is around 2.0 m, since the device was inluded in a vertical breakwater. However, as expected, the wave height is smaller using a single wave maker. Inside the OWC, Fig. 7a, mean free surface elevation also presents significant differences when using a single and multiple wave makers. Amplification factor, i.e. the ratio between the mean wave height measured inside the OWC and the incident wave height, was 3.0 and 4.6 using a single and a multiple wave makers, respectively, showing the importance, at the wave maker, of accurately cancel out the reflected waves that come from the OWC and vertical breakwater.

The high value of amplification factor, 4.6, was due to the fact that, for the present OWC design, a resonance occurs for wave periods around 7.5 s, as it was shown in Davyt et al. [24] and the device is not equipped with a turbine. It seems also that the amplification factor is higher due to the fact that the OWC was included in a vertical breakwater. The present design
can be associated to a harbor type OWC in a reflecting wall [25] which means that the dynamics of the harbor and of the OWC were combined, increasing, in the present OWC configuration and wave conditions, the excitation inside the OWC and the amplification factor.

**Figure 5:** Time series of free surface for single active absorption wave maker at 4 gauges in front of the wave maker (gauges G01 to G04) (a) and 3 gauges in front of the vertical breakwater (gauges G12 to G14) (b)

**Figure 6:** Time series of free surface for multiple active absorption wave makers at 4 gauges in front of the wave maker (gauges G01 to G04) (a) and 3 gauges in front of the vertical breakwater (gauges G12 to G14) (b)
Figure 7: Time series of mean free surface inside the OWC (a) and free surface in front of the OWC (gauge G11) (b) for single and multiple active absorption wave makers

5 CONCLUSIONS

With the objective of studying numerically wave energy devices of OWC type, a numerical wave tank including active absorption was developed based on the RANS–VOF FLUENT® numerical model, which is based on the RANS equations and the VOF surface capturing approach. The methodology which takes into account the active absorption at single and multiple wave makers is developed for a quasi-3D and a fully-3D wave tank.

Application to a quasi-3D wave tank, i.e. a 2D flume when the width of the OWC chamber and the tank are the same, shows that active absorption is absolutely necessary to correctly modelling the wave-structure interaction and calculating accurately the pneumatic power of the device. Furthermore, the multiple active absorption wave makers implemented in FLUENT® is validated for a fully-3D wave tank for the case of the interaction between an incident regular wave, with a wave period of 7.5 s and a wave height of 1.0 m, and an OWC device with 5 m by 5 m located at the end of the wave tank and inserted in a vertical breakwater. Results show that the mean free surface elevation presents significant differences along the wave tank using a single or multiple active absorption wave makers with amplification factors inside the OWC varying from 3.0 to 4.6, when using a single and multiple wave makers, respectively.

The implemented methodologies enable using the RANS-VOF model FLUENT® for modelling a fully 3D numerical wave tank and obtaining long time series for accurate analysis. The first results presented in this paper show the ability of the fully-3D numerical wave tank for modelling OWC devices accurately. Future works include mainly: completing the validation of the 3D numerical wave tank for a range of wave periods between 4 to 12 s, verifying the performance of the active absorption; taking into account the turbine effect, considering the typical Wells turbine curves or impulse turbine; and taking into account the
pressure air control inside the air chamber by a relief valve.

ACKNOWLEDGEMENTS

Paulo Teixeira acknowledges Engineering School and Oceanic Engineering Post-graduation of Universidade Federal do Rio Grande for the support. Eric Didier thanks the support of Fundação para a Ciência e a Tecnologia – FCT (project SFRH/BPD/97343/2013) and Unidade de Investigação e Desenvolvimento em Engenharia Mecânica e Industrial – UNIDEMI (project UID/EMS/00667/2013).

REFERENCES


AN EXPERIMENTAL STUDY OF THE HYDRODYNAMIC BEHAVIOR OF A TLP PLATFORM FOR A 5MW WIND TURBINE WITH OWC DEVICES

GEORGIOS M. KATSOUNIS*, STYLIANOS POLYZOS* AND SPYRIDON A. MAVRAKOS†

* Laboratory for Ship and Marine Hydrodynamics (LSMH) National Technical University of Athens (NTUA) 9 Heroon Polytechniou str., Zografos GR15773, Athens, Greece email: katsage@mail.ntua.gr, spolyzos@mail.ntua.gr

† Laboratory for Floating Structures and Mooring Systems (LFSMS) National Technical University of Athens (NTUA) 9 Heroon Polytechniou str., Zografos GR15773, Athens, Greece email: mavrakos@naval.ntua.gr

Hellenic Center for Marine Research, Director and President GR19013,Anavyssos, Greece email: mavrakos@hcmr.gr

Key words: Floating wind turbine, Tension leg platform, Wave energy converter, Oscillating water column, OWC, Wave Tank experiments

Abstract. An experimental study of the hydrodynamic behavior of a Tension-Leg Platform (TLP) for a 5MW Wind Turbine, featuring Wave Energy Converter (WEC) devices of the Oscillating Water Column type is presented. The examined triangular platform includes three vertical cylinders at the corners, providing the required buoyancy, each of them surrounded by a thin skirt, open at its lower end, forming the OWC chamber. A central vertical cylinder is included for the wind turbine installation. All cylinders are structurally connected with cylindrical bracing. The hydrodynamic response of the platform in the surge direction is experimentally verified, together with the resulting pressures and air fluxes inside the OWC chamber and the dynamic tensions in the lines of the mooring system.

1 INTRODUCTION

The development of the offshore wind industry is inevitably linked to the challenge of installation of wind turbines at water depths above the range of 40-70 meters, an area where fixed structures can’t be economically installed and operated. For such depths, several designs have been proposed (spar buoys, floaters, semisubmersibles and tension leg platforms), many projects are in the final design stage or in the experimental testing of scaled down prototypes, while there are also some pilot full scale developments [1].

It is evident that the overall cost of a floating installation is increased, as compared against
the fixed installations. Thus, any development in the design which results in an augmented harvesting of the available ocean renewable energy has obvious economic advantages. To this end, the combination on a single floating platform of a wind turbine and wave energy converters results in an obvious increase of the captured energy and, also, have both economic and operational advantages [2]. The analysis of the performance of the resulting system requires the simulation of the coupled operation of the wind turbine, the floating platform [3,4] and the OWC device [5,6]. An experimental investigation of the hydrodynamic response of the platform can produce valuable data, useful in the validation of the results of pertinent numerical analyses.

2 FLOATING PLATFORM DESCRIPTION

The examined triangular platform forms a floating basis for the installation of a 5MW NREL offshore wind turbine. The design includes three vertical cylinders at the corners, providing the required buoyancy. Each of them is surrounded by a thin skirt, open at its lower end, forming the OWC chamber. A central vertical cylinder is included in the arrangement for the installation of the wind turbine. All cylinders of the hull of the platform are structurally connected and supported by cylindrical bracing. The design is shown schematically in the following graph, where the domes of the OWC chambers have been removed to give access to the cylinders.
The mooring of the platform follows the tension leg concept. Three wire legs connect the bottom of the corner cylinders to the sea bed foundation, providing also the required pretension for the stabilization of the platform and for station keeping.

The weight groups of the platform and the tendon characteristics are summarised in the following table:

<table>
<thead>
<tr>
<th>Platform particulars</th>
</tr>
</thead>
<tbody>
<tr>
<td>Platform mass</td>
</tr>
<tr>
<td>Wind turbine and tower</td>
</tr>
<tr>
<td>Displacement</td>
</tr>
<tr>
<td>Vertical center of gravity of the platform (below sea level)</td>
</tr>
<tr>
<td>Nominal Tendon pretension (each)</td>
</tr>
<tr>
<td>Tendon diameter</td>
</tr>
<tr>
<td>Water depth</td>
</tr>
</tbody>
</table>

Under the action of the incident waves, the water contained between each cylinder and the corresponding skirt, acts as an oscillating water column and the resulting flux of air can be converted into electric energy by means of an air turbine. The hydrodynamic behavior of such an arrangement is quite complex, since the incident waves, the mooring, the motion of the platform, the behavior of the water column and the effect of the air turbine, are all interrelated and affect the ability of the system to convert wave energy into electricity.

In order to examine the behavior of such a system, an extensive set of experiments on a scaled down model of the platform (scale 1:40), were conducted in the wave tank of the Laboratory for Ship and Marine Hydrodynamics (LSMH) of the National Technical University of Athens (NTUA). The Froude scaling law was followed for the modelling of the wave environment. In the following paragraphs, the experimental setup and instrumentation is discussed, along with the modeling for the wind turbine and the wave energy converter.

Experiments were conducted for a wide range of incident waves, both harmonic and irregular, corresponding to the sea-states a TLP is expected to encounter in the Aegean Sea. In this work, the results of the above experimental study will be presented, in terms of the TLP motions in the surge x-direction under the action of harmonic waves, the loads on the mooring tendons, the wind turbine tower loads at its mounting location on the floating platform and the flow characteristics (flux and pressure) inside the WEC devices.

3 MEASURING SYSTEM

The response of the platform subjected to the wave action was recorded by an optical system. An array of four digital cameras was arranged at the side of the wave tank, to record the motion of special optical targets placed at various locations on the platform and on the tower of the wind turbine. The recorded frames were then analysed by optical recognition software, providing time histories of the motion.

The amplitudes of the waves generated by the wave maker of the tank were measured by two standard wave probes of wire type, one located near the wave maker while the other located in front of the platform. For the measurement of the internal water surface inside the
OWC device, three wave probes were used, located radially in the toroidal space of the OWC air chamber, spaced 120° apart. The elevation of the internal surface was obtained on the basis of these elevation measurements, assuming a flat shape for the internal surface and considering also that the motions of the TLP platform are mainly horizontal, due to the large amount of the pretension of the TLP system. The air volume flux was then computed by time differentiating the above measurements, taking into account the area of the OWC net cross section. All signals were sampled at a rate of 100Hz and subjected subsequently in digital filtering and recording.

The time histories of the pressures inside the air chamber of the OWC device were measured by three pressure transducers, two of them located on the dome of the chamber of the front cylinder, while the third sensing the outside pressure.

For the measurement of the exerted loads on the base of the tower of the wind turbine a 6-dof load cell was inserted, between the tower base and the platform. Furthermore, the accelerations of the platform were captured by accelerometers installed on the platform deck and along the wind turbine tower.

For the modeling of the static thrust of the wind turbine, two small thrusters were installed at the nacelle level and were calibrated to produce the required thrust.

Finally, three underwater load cells were inserted in the tendon lines, at their bottom end, for the measurement of the static pretension and dynamic tension of the mooring legs.

4 NATURAL FREQUENCIES IN SURGE MOTION

4.1 Stiffness in surge motion

The stiffness provided by the pretension or the tendons of the mooring system was measured by imposing specific loads along the surge –x direction and measuring the resulting offset of the platform. In this way, a value of 214 N/m was obtained, which corresponds to 342,4 kN/m for the full scale configuration.

4.2 Natural frequencies and damping coefficient

Free decay tests were carried out for the measurement of the natural frequency of the platform in surge motion. An initial displacement was imposed along the surge degree of freedom and the platform was then released to perform free decaying oscillations. A time history, representative of the decaying oscillations, is presented in figure 2. The measurements were analyzed in order to obtain the natural frequency and the damping parameter of the oscillation. The results for the frequency are summarized in Table 2, where it is shown that the surge natural frequency is 0.864 rad/sec (model scale), corresponding to 0.137 rad/sec in full scale.

The critical damping ratio in surge motion, as calculated on the basis of the logarithmic amplitude decay of the time histories, is presented in figure 3 (model scale), plotted against the amplitude of the oscillation. An increasing trend of the damping ratio with respect to the amplitude of oscillation can be concluded. For infinitesimal amplitudes the critical damping ration is obtained as 0.0307 for the model scale.
Table 2: Frequencies of free surge oscillations

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Model scale</td>
<td>Model scale</td>
<td>Full scale</td>
</tr>
<tr>
<td>27</td>
<td>7.10</td>
<td>0.885</td>
<td>0.140</td>
</tr>
<tr>
<td>50</td>
<td>7.29</td>
<td>0.862</td>
<td>0.136</td>
</tr>
<tr>
<td>74</td>
<td>7.30</td>
<td>0.861</td>
<td>0.136</td>
</tr>
<tr>
<td>120</td>
<td>7.34</td>
<td>0.856</td>
<td>0.135</td>
</tr>
<tr>
<td>119</td>
<td>7.33</td>
<td>0.857</td>
<td>0.136</td>
</tr>
<tr>
<td>Average</td>
<td>7.27</td>
<td>0.864</td>
<td>0.137</td>
</tr>
</tbody>
</table>

5 HYDRODYNAMIC RESPONSE

The platform was subjected to the action of harmonic wave trains, produced by the wave maker of the basin. The calculated non dimensional linear response amplitude operators (RAOs) for the surge motion of the platform are presented in figure 4.
Figure 5 presents the response operators for the induced accelerations at the base of the tower of the wind turbine, while Figure 6 depicts the corresponding coefficients for the bending moments at the same location. A peak of the above responses in the vicinity of 1.27 Hz (1.26 rad/sec full scale) is remarkable. This may result in a heavy loading of the tower of the air turbine, affecting also the fatigue strength of the structure.

![Figure 5: Linear acceleration at tower base](image)

![Figure 6: Bending moments at tower base](image)

### 6 MEASUREMENTS OF THE OWC PARAMETERS

The parameters that are relevant to the performance of the OWC device are the pressure difference (drop) between the air chamber and the outside space and the volumetric flow rate, passing through the air turbine of the device. Usually, a Wells turbine is used to convert the energy of the air flow to electricity. This is a bidirectional device, designed for directional changing air flows, like the ones produced in the air chamber of the OWC under the action of the oscillating water surface, due to the wave action. The integration of a scaled down model of a Wells air turbine in the presented model of the TLP platform was outside the scope of the work presented herein. Thus, an equivalent model for the simulation of the effect of the turbine on the air flow was necessary. For this purpose, an orifice (small diaphragm with a hole) was formed at the top of the conical domes of the OWC air chambers. Tests were carried out with the orifice hole having several diameters, namely 20, 40 and 50mm, since the effect of the orifice on the pressure depends on the value of the diameter. In general, the effect of a Wells turbine is approximated, in pertinent numerical models, by a linearised relationship between the pressure drop and the corresponding volumetric flow rate:

\[
\Delta p = \Lambda Q
\]

Where \(\Delta p\) is the pressure drop, \(Q\) the volumetric flow rate of the air, assumed incompressible, and \(\Lambda\) a device parameter, depended generally on the diameter, the rotational speed, and the specific particulars of the air turbine. The selection of the \(\Lambda\) value obviously affects the operation of the device, thus it should be subjected to optimization. Such a procedure is outside the scope of the presented work, as already mentioned, since no scaled down model of the air turbine was available. Following the approach with the orifice usage, an equivalent linearized relationship between pressure drop and volumetric flow rate can be established, by
linearising the nonlinear relation, pertinent to the orifice action. Indeed, the volumetric flow rate is related to the pressure drop by a general equation in the form of:

\[ Q = C_f A_0 \sqrt{\frac{\Delta p}{\rho}} \]  

(2)

where \( \rho \) the air density, \( A_0 \) the orifice area and \( C_f \) the orifice parameter, usually defined experimentally. Solving for \( \Delta p \) it can be obtained:

\[ \Delta p = \frac{1}{2} \rho \frac{1}{C_f A_0^2} Q^2 \]  

(3)

and, by considering bidirectional flows:

\[ \Delta p = C_e Q |Q| \]  

(4)

where \( C_e \) an equivalent coefficient, to be determined experimentally.

In the presented experimental work the instantaneous pressure drops was measured by the pressure transducers of the measuring system, while the volumetric flow rates were computed on the basis of the wave probe readings inside the air chamber. In the following figure the instantaneous pressures are plotted against the volumetric flow rates, in order to obtain the \( C_e \) values. Many individual curves are over-plotted in the figure, obtained from many experiments, with various wave amplitudes. The nonlinear character of the relation between the pressure and the flow rate is evident. However, an equivalent linearization can in principle be obtained, by defining a linear regression on a specific range of the pressures, or flows, of interest.

![Figure 7: Pressure drop to flow rate relation](image)

An approximate curve can be fitted to the above experimental data:

- **Orifice diameter 50mm, Flow rates up to 0.026 m\(^3\)/sec:**
  - **Compression phase:** \( \Delta p = 3 \times 10^3 Q |Q| \) \( \Delta p \) [mbar], \( Q \) [m\(^3\)/sec]
  - **Suction phase:** \( \Delta p = 4 \times 10^3 Q |Q| \)
Similar curves were obtained also for the orifice diameters of 40 and 20mm, resulting in the following estimations:

**Orifice diameter 40mm, Flow rates up to 0.025 m$^3$/sec:**

\[
\text{Compression phase: } \Delta p = 7 \times 10^3 \frac{Q}{|Q|} \quad \text{mbar}, \frac{Q}{m^3/sec}
\]

\[
\text{Suction phase: } \Delta p = 8 \times 10^3 \frac{Q}{|Q|}
\]

Orifice diameter 20mm, Flow rates up to 0.006 m$^3$/sec:

\[
\text{Compression phase: } \Delta p = 100 \times 10^3 \frac{Q}{|Q|} \quad \text{mbar}, \frac{Q}{m^3/sec}
\]

\[
\text{Suction phase: } \Delta p = 80 \times 10^3 \frac{Q}{|Q|}
\]

The pressure drops in the front cylinder, plotted against the frequency of the incoming waves, are shown in the following figures. The values are given as linearised RAO’s and were based on tests with harmonic waves having various amplitudes. A scattering of the values can be observed, maybe due to the aforementioned nonlinear character of the orifice equation, which affects the pressure formation.

**Figure 8:** OWC chamber pressures, Orifice D=20mm  
**Figure 9:** OWC chamber pressures, Orifice D=40mm  
**Figure 10:** OWC chamber pressures, Orifice D=50mm
7 DYNAMIC TENSION ON THE MOORING LINES

The dynamic tension exerted on the mooring lines of the TLP platform seems dominated by the pressure formation inside the chambers of the OWC devices. The following figures present the dynamic tension of the mooring line of the front cylinder, as linearised RAO’s. The same observation regarding the scatter of the values can also be made here, like in the case of the pressure drop.

![Figure 11: Tendon dynamic tension, Orif. D=20mm](image1)

![Figure 12: Tendon dynamic tension, Orif. D=40mm](image2)

![Figure 13: Tendon dynamic tension, Orif. D=50mm](image3)

12 CONCLUSIONS
- An experimental study of the hydrodynamic behaviour of a TLP platform for the installation of a 5MW NREL wind turbine and three OWC devices is presented.
- Data for the natural frequency in surge motion and response amplitude operators for
surge motion, pressures inside the OWC chamber and dynamic tensions in the tendons were experimentally obtained.

- The relation between the chamber pressure and the volumetric flow rate in the OWC chamber was presented for several simulations of the effect of the air turbine, through the use of an orifice device.
- The presented data can be useful in the validation of the results of numerical simulations of motion and performance of this complex system.

REFERENCES


Assessment of the parametric roll behaviour of a point absorber wave energy conversion device in irregular waves

VII International Conference on Computational Methods in Marine Engineering
MARINE 2017
M. Visonneau, P. Queutey and D. Le Touzé (Eds)

ASSESSMENT OF THE PARAMETRIC ROLL BEHAVIOUR OF A POINT ABSORBER WAVE ENERGY CONVERSION DEVICE IN IRREGULAR WAVES.

Craig Meskell* & Kevin R. Tarrant
*Dept. of Mechanical and Manufacturing Engineering,
Trinity College, Dublin
Ireland
e-mail: cmeskell@tcd.ie

Key words: Wave Energy Conversion, Parametric Resonance, Irregular Waves

Abstract. The response of the Wavebob wave energy conversion device has investigated numerically in long crested irregular waves generated using the JONSWAP wave spectrum. Only long crested waves are considered as the roll mode will not be directly excited by the incident wave since the device is axisymmetric. As a result all roll response is due to mode coupling associated with parametric roll. The simulations were conducted at 17th scale to allow direct comparison with the regular wave case. The simulations were conducted for a significant wave height of 0.2m and a range of wave peak periods. The response of the device is simulated at each condition for 2000 seconds of simulated time. In all conditions, it was found that there was a significant roll response at the roll natural frequency for all wave excitation frequencies, including excitation frequencies outside the main zones of parametric resonance, even though the roll mode is not directly excited in long crested waves. It is concluded that there will always be some level of parametric excitation in irregular waves due to broadband excitation. At the tuning factor of 2, there are sudden bursts of roll motion due to parametric roll, however the large roll amplitudes are not sustained as they were in regular waves with parametric resonance.

1 INTRODUCTION

Ocean Waves as a renewable energy source offer a relatively high energy density and limited impact on competing land uses (e.g. agriculture, tourism). The estimated resource is significant. For example, it has been estimated that the North-Eastern Atlantic could be as much as 290GW [1]. However, the technology to capture and convert the energy is still immature, as evidenced by the numerous approaches proposed and under development. Falcão estimated there were over 1000 patents on converting wave energy. Detailed reviews of these technologies have been presented by, for example Drew et al. [2] and Falcão [3].

The particular wave energy converter (WEC) under consideration is called the Wavebob. This WEC is an axisymmetric, self-reacting point absorber with power take-off only in the heave mode. It consists of two concentric floating buoys: a torus and a float-neck-tank (FNT).
The FNT is a long, heavy, largely cylindrical component. The FNT pierces the sea surface in the centre of the torus. The two floats are mechanically constrained so that the only possible relative motion between them is in the heave direction. As the masses and hydrodynamic properties of the two floats are quite different, an incident wave train excites a relative heave motion between the two floats. The Torus is effectively a wave follower, while the FNT is the reaction mass. In scale models, this arrangement has been found experimentally to experience parametric roll, at least in regular waves. It is well known that floating bodies, such as self-reacting WECs, may exhibit parametric resonance. This condition occurs when the incident wave frequency is approximately twice the pitch or roll frequency. The coupled fluid structure system exhibits coupling of at least two degrees of freedom, and the response amplitude may be large even in degrees of freedom which are not excited directly. While survivability may be an issue ultimately, for the Wavebob, the onset of parametric roll causes a dramatic reduction in the relative heave, and hence power take-off, as energy is transferred to the roll mode. Parametric roll is not unique to Wavebob. Spar like structures in general will be susceptible and there is evidence that other WEC devices may exhibit this type of behaviour [4]. The large amplitude motions associated with parametric roll necessitate a non-linear model of the system. A numerical scheme for modelling this situation has previously been described [5], and has been demonstrated for the Wavebob in regular waves. The results were validated against 17\textsuperscript{th} scale wave tank test results, and found to agree well.

An irregular wave analysis is desirable important to ascertain whether or not a point absorber is susceptible to parametric motion in more realistic seas, and if so, if the behaviour of the phenomenon is similar to that which occurs in regular waves. Parametric roll has been proven to be a dangerous phenomenon for several categories of ships occurring in realistic sea states, as outlined by France et al. [6]. In particular, it is to be noted that the occurrence of parametric roll of ships in irregular seas is much more dangerous and subtle than the same phenomenon in regular seas. This is due to the fact that when the parametric excitation is random, the build-up of roll can occur suddenly and abruptly after very long periods of quiescence, with a fast increase in the roll amplitude after only a few roll cycles [7].

2 NUMERICAL SCHEME

A time-domain nonlinear numerical model is used to investigate the dynamic response of the Wavebob. The pressure of the incident wave is integrated over the instantaneous wetted surface to obtain the nonlinear Froude-Krylov excitation force and the nonlinear hydrostatic restoring forces, while first order diffraction-radiation forces are computed by a linear potential flow formulation. These four components have been calculated directly.

In addition to these fluid forces, a non-linear viscous drag force is included using a form similar to the Morison equation. The drag coefficient matrix has been estimated experimentally with free decay tests in quiescent water. The mooring is treated as a linear spring. The power take off (PTO) provides a linear damping proportional to the relative heave motion only. Details of the modeling approach and numerical scheme used can be found in [5].
3 IRREGULAR WAVE MODEL

Assessment of the motion response of the device in irregular waves requires a suitable model of the real ocean environment. Irregular waves can be classified as either long crested or short crested based upon the direction of wave propagation. If the irregularities of the observed waves are only in the dominant wind direction, so that there are mainly mono-directional wave crests with varying separation but remaining parallel to each other, the sea is referred to as a long crested irregular sea. Throughout this paper the sea state will be considered long crested. This has the advantage that only heave, pitch and surge are directly excited and so response in roll can be attributed to non-linear interaction.

A linear superposition of $N$ regular wave components, each having different amplitude, frequency, and phase angle is used to generate long crested waves. The wave elevation time series can therefore be represented as

$$\zeta_w(x, t) = \sum_{n=1}^{N} \zeta_{no} \cos(k_n x - \omega_n t + \theta_n) \quad (1)$$

where $\theta_n$ is the phase of component $n$, with amplitude $\zeta_{no}$ and wave number $k_n$. The equally spaced frequencies are given by $\omega_n = 2\pi n / T_H$, where $T_H$ is the length of the time history.

The energy per square meter of the sea surface of the $n$th wave component is

$$E = \frac{\rho g \zeta_{no}^2}{2} \quad (2)$$

The wave spectrum are represented by the symbol $S_\zeta(\omega_n)$ and is given by

$$S_\zeta(\omega_n) = \frac{\zeta_{no}^2}{2\delta\omega} \quad (3)$$

where $\delta\omega$ is the fixed interval between frequencies. Therefore, from Eqn. 2, the total energy per square meter of the wave system is equal to the total area enclosed by the wave spectrum multiplied by the factor $\rho g$.

The inverse transformation can also be made in order to generate a time history according to Eqn. 1 from a given wave spectrum. The amplitude of the $n$th component sinusoidal wave is given by

$$\zeta_{no} = \sqrt{2S_\zeta(\omega_n)\delta\omega} \quad (4)$$

Therefore, the time history of wave elevation is obtained by substituting Eqn. 4 into Eqn. 1 such that

$$\zeta_w(x, t) = \sum_{n=1}^{N} \sqrt{2S_\zeta(\omega_n)\delta\omega} \cdot \cos(k_n x - \omega_n t + \theta_n) \quad (5)$$

The wave spectrum used in the current work is the JONSWAP (Joint North Sea Wave Project) spectrum described by Hasselmann et al. [8]. A fully developed sea exists in conditions where there is sufficient fetch available, and the wind blows at a constant velocity for long enough such that the rate at which energy is absorbed by the waves will eventually be exactly balanced.
by the rate of energy dissipation. The waves in the JONSWAP spectrum thus continue to evolve with distance (or time). The spectral ordinate of the JONSWAP spectrum is defined as

$$S_{J\zeta}(\omega) = \frac{\alpha g^2}{\omega^5} \exp \left[ -1.25 \frac{\omega^4_p}{\omega^4} \right] \gamma^\alpha$$

(6)

where

$$a = \exp \left[ -\frac{(\omega - \omega_p)^2}{2\omega_p^2\sigma^2} \right]$$

and

$$\sigma = \begin{cases} 0.07 & \text{if } \omega \leq \omega_p \\ 0.09 & \text{if } \omega > \omega_p \end{cases}$$

The parameter $\gamma$ in Eqn. 6 is called the peak-enhancement factor, the effect of which is to increase the peak of the spectrum. The other parameters to define the sea state are the peak frequency, $\omega_p = 2\pi/T_p$, which is the frequency with the maximum value in the spectrum; $T_p$, which is the peak period of the sea state; and $H_s$, the significant wave height which is defined as the average height of the highest one-third wave peaks in a wave spectrum. The parameter $\alpha$ in Eqn. 6 is included to take into account the growth of the waves with distance.

The motion response spectrum for the translational motion (surge, sway and heave) is calculated by filtering the wave energy spectrum with the motion transfer function. The translational motion transfer function is defined as the motion amplitude divided by the wave amplitude in the particular degree of freedom for each wave frequency. The filtering procedure is achieved by multiplying the wave spectrum $S_{\zeta}(\omega)$ by the square of the translational motion transfer function. Note that in order to distinguish the wave spectrum from the motion response spectrum, the ordinates of the motion response spectrum are denoted by $S_{\xi_j}$, corresponding to the particular degree of freedom $\xi_j$. The motion response energy spectrum ordinate at each wave frequency for the translational degrees of freedom ($\xi_j, j = 1, 2, 3$) of a single body system is given by

$$S_{\xi_j}(\omega) = S_{\zeta}(\omega) \left( \frac{\xi_j}{\zeta_o(\omega)} \right)^2$$

(7)

A similar procedure is followed to calculate the motion response spectra for the angular motion responses (roll, pitch and yaw), except that the transfer functions are normalised by dividing by the wave slope amplitude $k\zeta_o$. Also, the wave slope spectrum $S_{\alpha}(\omega)$ is used instead of the wave energy spectrum $S_{\zeta}(\omega)$ for calculating the motion response spectrum in the case of the rotational modes. The wave slope spectrum is given by

$$S_{\alpha}(\omega) = \frac{\omega^4}{g^2} S_{\zeta}(\omega)$$

(8)

The motion response energy spectrum for the roll, pitch and yaw degrees of freedom ($\xi_j, j = 4, 5, 6$) of a single body system is then given by

$$S_{\xi_j}(\omega) = S_{\alpha}(\omega) \left( \frac{\xi_j}{k\zeta_o(\omega)} \right)^2$$

(9)
Table 1: Simulated conditions in long crested waves

<table>
<thead>
<tr>
<th>Sea State</th>
<th>$H_s$ [m]</th>
<th>$T_p$ [s]</th>
<th>$\omega_p$ [rad/s]</th>
<th>$\omega_p/\omega_4$</th>
<th>$B_{PTO}$ [Ns/m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.2</td>
<td>1.25</td>
<td>5.027</td>
<td>4.7059</td>
<td>5000</td>
</tr>
<tr>
<td>2</td>
<td>0.2</td>
<td>2.94</td>
<td>2.137</td>
<td>2</td>
<td>5000</td>
</tr>
<tr>
<td>3</td>
<td>0.2</td>
<td>3.33</td>
<td>1.887</td>
<td>1.7665</td>
<td>5000</td>
</tr>
</tbody>
</table>

where $k = \omega^2/g$ is the wavenumber. The variance, $m_{o_j}$, of the response motion in degree of freedom $j$ is obtained by integrating the motion energy spectrum according to

$$m_{o_j} = \int_{0}^{\infty} S_{\xi_j}(\omega) d\omega$$

(10)

The root-mean-square (RMS) of the motion in degree of freedom $j$ is then given as

$$\xi_{j_{rms}} = \sqrt{m_{o_j}}$$

(11)

4 RESULTS

The response of the Wavebob to three sea states are presented each with a significant wave height of $H_s = 0.2m$. Table 1 outlines the parameters associated with each condition. The tuning factor is also shown, which is defined for irregular waves as the ratio of the peak wave frequency to the roll natural frequency ($\omega_p/\omega_4$). This set of conditions were chosen as they correspond to a low period wave (Sea State 1), a wave at the parametric resonance tuning factor of 2 (Sea State 2) and a high period wave (Sea State 3).

The time traces for the relative heave, roll and pitch motions for the three sea states are presented in Figs. ?? respectively. The corresponding wave energy spectra, transfer functions and motion response energy spectra for the same conditions are shown in Figs. 4 - 6. Since only long crested waves are considered in this analysis, there is no direct wave excitation in the roll mode. Since there is no roll excitation, and hence no roll transfer function, only the roll motion response energy spectrum is presented for each test, which is calculated as the spectral density of the roll response.

4.1 Sea State 1

In Fig. 4(a) (Sea State 1), the frequency range of greatest wave energy content ($\omega_p = 5.027$ rad/s) is far from the heave natural frequency of 2.199 rad/s. For this reason, a small amount of the wave energy is transferred into first order heave motion, as observed from the relative heave transfer function and relative heave energy spectrum. Furthermore, as can be seen in Fig. 4(b) the frequency range of greatest wave energy content is far from the pitch natural frequency of 1.068 rad/s. Therefore, the pitch transfer function and pitch energy spectrum show almost zero motion, with most of the energy occurring at the pitch natural frequency. From Fig. 4(c), it is seen that the roll motion for this test shows a significant response, even though the roll is not being directly excited. The tuning factor of 4.7059 is far from the primary region of parametric resonance, however, as the system is being excited by broadband excitation, there will always be some level of parametric excitation at twice the roll natural frequency, which accounts for the roll response.
4.2 Sea State 2

For Sea State 2, at the parametric resonance tuning factor of $\omega_p/\omega_4 = 2$, the RMS of the relative heave is over six times greater than that of Sea State 1, since the peak of the wave energy spectrum for this test condition ($\omega_p = 2.137 \text{ rad/s}$) is located around the heave natural frequency of 2.199 rad/s. Therefore, a large amount of the wave energy is transferred to the heave motion, as shown in Fig. 5(a).

The corresponding time trace in Fig. 2 reveals evidence of parametric roll occurring, as seen from the sudden bursts of roll motion taking place, in particular at around 1400 seconds, where the roll angle reaches a maximum amplitude of .07 radians. The roll motion energy spectrum in Fig. 5(c), shows that the RMS of the roll is over twice the value it was in Sea State 1 at the lower period of 1.25s, whereas the maximum value of the spectral ordinate in the roll motion energy spectrum is over five times greater. In Fig. 5(b), it can be seen that the pitch motion RMS is almost the same as Fig.4(b). However, most of the energy for the pitch motion is again, at the pitch natural frequency.

A time-frequency analysis of Sea State 2 is shown in Figs. 7 to illustrate how the response spectrum of the different modes evolve over time. The dominant tuning factor for this condition is 2, and it can be seen that the roll motion responds mainly at it’s own natural frequency for the duration of the test. For the first 200s, the pitch motion is responding with a low amplitude, at both the wave excitation frequency and the pitch natural frequency. After 200s, the largest pitch response amplitude is at the pitch natural frequency, which is typical of parametric motion. The heave motion in this test is responding at the wave peak frequency ($\omega_p = 2.137 \text{ rad/s}$) and at the heave natural frequency of 2.199 rad/s.

4.3 Sea State 3

In the final condition there is a slight reduction in the relative heave compared to Sea State 1, however, the heave mode is still absorbing a large amount of the wave energy as shown in Fig. 6(a) as the wave energy peak frequency ($\omega_p = 1.887 \text{ rad/s}$) is still close to the heave natural frequency. Fig. 6(c) shows the roll RMS is approximately half the value it was in Sea State 1, as there is less frequency content at twice the roll natural frequency, and hence less parametric excitation occurring in this condition.

5 CONCLUSIONS

For the long crested sea states shown here, there is always a roll response at the roll natural frequency, with the greatest amplitudes occurring at the wave tuning factor of 2 (Sea State 2). In irregular waves, there is always going to be a significant level of roll response occurring, even though there is no direct wave excitation in the roll mode since there will always be frequency content occurring at twice the roll natural frequency due to the broadband nature of the excitation. This is different from the case in regular waves [5], where significant roll response was observed only in waves which had frequencies in the region of the tuning factors of 1 or 2. For waves outside these unstable zones, the roll response was almost zero. In the case of the pitch mode, it was seen that the dominant response in regular waves was close to the excitation frequency for all wave excitation frequencies, except at a frequency ratio of 2 when parametric motion occurred. In the case of long crested waves, most of the pitch motion energy
is concentrated at the pitch natural frequency, not just at the tuning factor of 2 but also at tuning factors of 4.7059 and 1.7665, as is seen in the roll mode.

It seems likely that since any realistic sea state with irregular waves will always have some level of excitation at twice the pitch/roll natural frequency due to broadband excitation, there will always be some level of parametric motion taking place. The non-linear coupling will therefore give rise to significant roll motion even in the absence of direct excitation.

6 ACKNOWLEDGEMENTS

The authors gratefully acknowledge the financial support of: Irish Research Council, Grant IRCSET-WAVEBOB-2010-01 for K. Tarrant; Science Foundation Ireland, Grant 15/SPP/E3125 for C. Meskell.

REFERENCES


Figure 1: Time series of relative heave, roll and pitch motion for Sea State 1 ($T_p = 1.25s$)

Figure 2: Time series of relative heave, roll and pitch motion for Sea State 2 ($T_p = 2.94s$)

Figure 3: Time series of relative heave, roll and pitch motion for Sea State 3 ($T_p = 3.33s$)
Figure 4: Calculation of motion energy spectra for (a) relative heave, (b) pitch and (c) roll for Sea State 1 $H_s = 0.2m$, $T_p = 1.25s$ and $B_{PTO} = 5000$ Ns/m
Figure 5: Calculation of motion energy spectra for (a) relative heave (b) pitch and (c) roll for Sea State 2 $H_s = 0.2m$, $T_p = 2.94s$ and $B_{PTO} = 5000$ Ns/m
Figure 6: Calculation of motion energy spectra for (a) relative heave (b) pitch and (c) roll for Sea State 3 $H_s = 0.2m$, $T_p = 3.33s$ and $B_{PTO} = 5000$ Ns/m
Figure 7: Time frequency analysis of response to Sea State 2 for (a) relative heave (b) roll and (c) pitch for $H_s = 0.2m$, $T_p = 2.94s$ and $B_{PTO} = 5000$ Ns/m
HYBRIDIZATION OF FINITE ELEMENT-BOUNDARY ELEMENT METHODS USING AN ABSORBING BOUNDARY CONDITION FOR VIBRO-ACOUSTIC UNDERWATER NOISE SIMULATIONS

MARINE 2017

N. ZERBIB†, K. BOUAYED†, J. LEFEVBRE†, M. ANCIANT†, L. MEBAREK†

† ESI Group, Center of Excellence Vibro-Acoustics
8, rue Clement Bayard
60200, Compiègne, France
email: nicolas.zerbib@esi-group.com, www.esi-group.com

Key words: Fluid Structure Interaction, Underwater, Added Mass, Sub-structuring, Finite Element Method, Boundary Element Method, MultiLevel Fast Multipole Method, Hybridization, Adaptive Absorbing Boundary Condition

Abstract. Sound propagation from industrial activities in underwater or estimation of the target strength of waterside security systems are considered immensely important by many scientists for both regulators of development projects or military aspects. This paper presents a comparison of numerical methods used to model large scale acoustic coupled fluid structure interaction underwater problems. Concerning the mechanical behavior of the structure, it is absolutely essential during the computation of the modal basis to take into account the added mass effect of the heavy fluid, the water, around the structure. In that work, the added mass matrix is evaluated by a Boundary Element Method and the modal basis is computed by a sub-structuring algorithm to deal with both large number of degrees of freedom and modes. On the acoustic point of view, this article presents an efficient way to deal with very complex and large scale underwater target. This method is competing against standard Perfectly Match Layers (PML), Infinite Element Method (IEM), standard Boundary Element Method (BEM) and more recently the coupled MultiLevel Fast Multipole Method (MLFMM). Reducing considerably both computational time and RAM requirement keeping a very good accuracy, this approach hybridizes advantages of FEM, BEM and MLFMM methods through a domain decomposition technic using an Adaptive Absorbing Boundary Condition (AABC). Some numerical results are presented to present the capabilities of that approach on academic cases but also on more industrial applications.

1 INTRODUCTION

Predicting the vibro-acoustic responses of thin shell structures in contact with an unbounded fluid is important in various engineering applications including underwater vehicles and submarines. Low frequency vibration modes of a thin shell can be easily excited by external forces for example, which may result in a high level of radiated noise. Identifying the modal contributions to the sound radiation of shell structures is useful to reduce the noise by refining the design of the structure. For an elastic shell in air, the structural and acoustic responses can be subsequently solved. However, for the case of a shell immersed in water where the fluid
impedance is comparable to that of the shell, the fluid-structure interaction is strongly coupled and the structural and acoustic responses have to be simultaneously solved.

1.1 The Target Strength

An acoustic scattering signature is the Target Strength (TS) of an object, Eq (1), that has been impinged on a broad band of frequencies and, for each frequency, over a broad range of aspect angles. When the obstacle is impinged on a plane wave (wave fronts are planar, rather than curved, as occurs when the sound source is far away from the object), the TS is defined as

$$S(\alpha, \beta) = \lim_{r \to \infty} 10 \log_{10} \left( \frac{\|r - r_0\|^2 p_s^2(r, \alpha, \beta)}{p_i^2(r_0, \alpha_i, \beta_i)} \right)$$

(1)

with $p_s(r, \alpha, \beta)$ denoting the scattered acoustic pressure at range $r$ and $p_i(r_0, \alpha_i, \beta_i)$ the incident field at the geometric location of the center of the target, which is conveniently set to the origin i.e. $r_0 = 0$. This is evaluated in the farfield i.e. $r \gg L^2/\lambda$, with $L$ denoting the characteristic length of the target and $\lambda$ the acoustic wavelength. In the farfield TS is not dependent on range $r$, $\alpha$ is the aspect angle and $\beta$ the elevation angle as depicted in Fig. 1. For monostatic TS, the incident and "scattered" angles, respectively in light green and in orange as depicted Fig. 1, are equal i.e. $(\alpha_i = \alpha, \beta_i = \beta)$. The aspect angle $\alpha$ is usually the azimuthal angle, which is a horizontal angle about the vertical to the ocean bottom. The elevation angle $\beta$ is the angle between the horizontal plane and the line of sight, measured in the vertical plane. The elevation angle $\beta$ is positive above the horizon (0° elevation angle) and negative below the horizon.

![Figure 1: The General Problem of Scattering Strength.](image)

In order to have access to the acoustic scattering signature of their system, one could perform experiments on actual objects. But experiments are expensive and time consuming so only a few can be performed and one cannot perform experiments on unavailable objects or environments. Computers, however, can model virtually any object/environment scenario of interest, including non-existent scenarios. The cost of computer resources per model is negligible compared to that of a real underwater experiment and often faster by orders or magnitude, sometimes enabling hundreds or thousands of templates to be computed in the same time as performing one underwater experiment. There is clearly a need for a computer simulation system that is both high-fidelity and computationally fast. The principle challenges for developing such a system are as follows:
- Multiscale spatially: From small details in the objects (cm) to large distances in the ocean (km).
- Broadband: A five-octave range, \( kL = 1 \) to \( kL = 1000 \) where \( kL \) is dimensionless frequency, \( k \) is the wavenumber \( (k = 2\pi/\lambda) \), \( \lambda \) is the wavenumber of the impinging plane wave and \( L \) is the characteristic length of the target.
- A need for extraordinarily high computational efficiency: One acoustic signature template requires sweeping typically over several hundred frequencies, and, for each frequency, several hundred aspect angles, requiring \( O(10^5) \) 3-D models.

In the field of computational structural acoustics, the problem of efficiently and as quickly as possible modelling the acoustic field in large exterior domains has remained a difficult challenge for over a quarter century. The coupled Finite Element Method/Boundary Element Method is a very powerful and the most popular tool for computing the vibro-acoustic responses of fluid-loaded structures [4-8]. The FEM method is generally employed to describe the dynamic behavior of the structure whereas the BEM method is used to represent the fluid domain and predict the acoustic responses. This approach produces a deterministic prediction of acceleration, force and stress in the structure. It reveals itself very accurate at low frequency and is still very meaningful across all the spectrum of interest of the vibro-acoustic response. Because BEM matrices tend to be fully populated rather than banded as in FEM, the computational effort to assemble the equations and solve the linear system can be significant. In BEM computations, the CPU time scales with the number of degrees-of-freedom \( N \) as \( O(N^3) \) and the memory required as \( O(N^2) \). Numerical models with typical sizes, analyzed within the frequencies of interest, require long computation times. This precludes the recurrent application of the FEM/BEM approach as new versions of the model design emerge since the early phases, as well as carrying out multiple iterations for optimizing it. Moreover, a modelling overhead is imposed in which the BEM mesh is created based on system-level FEM meshes but made coarser in order to yielding acceptable computation times.

More recently, the coupled Multilevel Fast Multipole Method (MLFMM) [reference ESA/ESTEC] enables to considerably accelerate FEM-BEM computations, without loss of accuracy. It consists of an iterative multi-scale hierarchical clustering of the acoustic sources forming the BEM mesh and allows for a drastic simplification of the long distance interactions. The MLFMM reduces scaling laws with respect to memory and run-time from \( O(N^3) \) and \( O(N^2) \) in the standard BEM down to \( O(N_{\text{iter}} \times N \times \log^2(N)) \) and \( N \times \log^2(N) \). \( N_{\text{iter}} \) is the number of iterations in the solver which can be kept sufficiently low for most of the applications of interest. Nevertheless, this approach has also several weaknesses. The main constraint of this method is it relies on an iterative solver. The convergence can be very slow (or not converged at all) because of the complexity of the physics even by using some advanced preconditionner technics like ILUT or SPAI, the quality and the number of nodes of the mesh and the frequency. Even if some preconditionner techniques have been developed, the number of iterations \( N_{\text{iter}}^{\text{FMM}} \) can be very large or infinite for some cases. Moreover, as with every iterative solver, it is not really adapted to deal with a very large number of load cases even if some techniques have been developed to treat this kind of application. In conclusion, this method is not really adapted to compute monostatic TS needing a very large number of plane waves for very complex structure.
like a submarine for example. Consequently, a numerical method that is applicable at all frequency and dimension ranges of target is proposed in this paper for fully coupled vibro-acoustic problems keeping advantages of all the standard methods by hybridization of those method by a Domain Decomposition Method (DDM) with overlapping.

2 THE GENERAL COUPLED VIBRO-ACOUSTIC FLUID-STRUCTURE PROBLEM

We consider an elastic structure $\Omega_s$ located in a domain $\Omega_f$ filled by a perfect fluid. We define by $\Sigma_s = \partial \Omega_s$ and $\Sigma_f = \partial \Omega_f$ their respective surfaces. We denote by $n$ the surface unit normal vector oriented inward the fluid domain $\Omega_f$. This elastic structure can be a submarine for example as illustrated on Fig. 2 or a ship hull. This structure can be submitted to an acoustic incident pressure field $p^{inc}$ or a mechanical force $f$.

![Figure 2: Elastic structure $\Omega_s$ located in a domain $\Omega_f$ filled by a perfect fluid.](image)

The coupled vibro-acoustic fluid-structure problem consists then in solving simultaneously two physical problems:

1. In the first one, a pressure field $p$ is imposed by the fluid on the surface $\Sigma_s$ of the elastic structure
2. In the second one, a displacement field $w$ is imposed on the surface $\Sigma_f$ in the fluid domain from the structure.

The structure behavior obeys to the well-known linear elasticity laws in the harmonic domain at the $\omega$ pulsation for the conservation of the quantity of movement:

$$
\sigma_{ij} - \rho_s \omega^2 w = 0 \quad \text{in} \, \Omega_s
$$

where $\rho_s$ is the volumic mass of the structure, $w$ the displacement and $\sigma_{ij}$ the tensor of stress.

We define by the compatibility relation the tensor of constrain as follow:

$$
\varepsilon_{kl} = \frac{1}{2}(w_{kl} + w_{lk})
$$

We also define the behavior law in isotropic linear elasticity by

$$
\sigma_{ij} = C_{ijkl} \varepsilon_{kl}
$$
where the elasticity modules verify the identities $C_{ijkl} = C_{klij} = C_{ijkl} = C_{jilk}$ ($C$ is the elasticity tensor). Finally, the boundary condition on the surface $\Sigma_s$ of the elastic structure is defined by

$$\sigma_{ij}n_j = f_i$$

Concerning the fluid, we consider the propagation of time harmonic acoustic waves in a homogeneous isotropic acoustic medium (which can be either finite or infinite) as described by the well-known Helmholtz equation:

$$\nabla^2 p(x) + k^2 p(x) = 0 \quad \text{in } \Omega_f$$

where $p$ is the pressure field in the fluid domain, $k = \omega/c$ is the wave number, $\omega$ is the angular frequency and $c$ is the wave speed in the acoustic medium whose density is denoted by $\rho_f$. For the coupling part between these two physics at the fluid-structure interface, the fluid being non-viscous, it does not adhere to the surface, which leads to:

- Continuity to the normal constrains:
  $$\sigma_{ij} \cdot n = -p\delta_{ij} \cdot n \quad \text{on } \Sigma_s$$

- Continuity of the normal displacements:
  $$\frac{\partial p}{\partial n} = \rho_f \omega^2 w \quad \text{on } \Sigma_f$$

Finally, the formulation for the coupled vibro-acoustic fluid-structure problem, expressed in terms of displacements for the structure and pressure for the fluid is

$$\begin{cases}
\sigma_{ij} - \rho_s \omega^2 w = 0 & \text{in } \Omega_s \\
\nabla^2 p(x) + k^2 p(x) = 0 & \text{in } \Omega_f \\
\sigma_{ij} \cdot n = -p\delta_{ij} \cdot n & \text{on } \Sigma_s \\
\frac{\partial p}{\partial n} = \rho_f \omega^2 w & \text{on } \Sigma_f \\
\lim_{r \to \infty} r\frac{\partial}{\partial r} - ikp_s = 0
\end{cases}$$

To obtain a unique solution, it is necessary to close the system by considering a radiation boundary condition to take into account the pressure of the scattered wave $p_s = p - p^{inc}$ vanishes far away from the object, in the so-called far field. It is also possible to deal with an interior fluid domain different from the exterior one, filled by air for example, by splitting the $\Omega_f$ into two different medium characteristics in the second equation of the global linear system Eq. 9.

## 4 FINITE ELEMENT METHOD FOR THE ELASTIC STRUCTURE

Using the equation of elastodynamic of the structure Eq. 2, the boundary condition Eq. 5 and the coupling condition Eq. 7, we write the variational formulation of the structure expressed in terms of displacement field:

$$K_{ss}(\bar{w}, \bar{w}') - \omega^2 M_{ss}(\bar{w}, \bar{w}') + C_{ss}(p, \bar{w}') = F_{ss}(\bar{f}, \bar{w}')$$
where \( \mathbf{w}' = \overline{\mathbf{w}}' \cdot \mathbf{n} \) is the normal component of the displacement vector. \( \mathbf{K}_{s} \) and \( \mathbf{M}_{s} \) are respectively the rigidity and the mass matrices defined by:

\[
\mathbf{K}_{s}(\mathbf{w}, \mathbf{w}') = \int_{\Omega} \sigma_{ij}(\overline{\mathbf{w}}). \varepsilon_{ij}(\mathbf{w}') d\Omega
\]

\[
\mathbf{M}_{s}(\mathbf{w}, \mathbf{w}') = \int_{\Omega} \rho \overline{\mathbf{w}}. \overline{\mathbf{w}}' d\Omega
\]

\( \mathbf{F}_{s} \) is the operator related to the mechanical forces defined by

\[
\mathbf{F}_{s}(\mathbf{f}, \mathbf{w}') = \int_{\Sigma} \overline{\mathbf{f}} . \overline{\mathbf{w}}' d\Sigma
\]

and \( \mathbf{C}_{s} \) is related to the coupling matrix with the acoustic defined by

\[
\mathbf{C}_{s}(\mathbf{p}, \mathbf{w}') = \int_{\Sigma} \mathbf{p} . (\overline{\mathbf{w}}'. \mathbf{n}) d\Sigma
\]

### 3.1 Modal resolution and Added Mass Matrix

To solve the Eq. 10, usually, a modal analysis of the structure can be used to reduce the CPU time instead of using the physical unknowns, i.e. displacements at every nodes of the mesh of the structure. It consists in computing the modes \( \Psi_{r} \) and their corresponding frequencies \( \omega_{r} \) of the structure in a free-free status with no damping by solving:

\[
\det(\mathbf{K}_{s} - \omega^2 \mathbf{M}_{s}) = 0
\]

Both mass and rigidity matrices can be then reduced on that modal basis \( \{\Psi^{i}_{r}\}_{i=1,...,m} \), \( m \) standing for the number of modes used to represent the dynamic behavior of the structure as follows:

\[
\mathbf{K}_{r} = \{\Psi^{i}_{r}\}^{T} \mathbf{K}_{s} \{\Psi^{i}_{r}\}
\]

\[
\mathbf{M}_{r} = \{\Psi^{i}_{r}\}^{T} \mathbf{M}_{s} \{\Psi^{i}_{r}\}
\]

The generalized forces are defined by:

\[
\mathbf{F}_{r} = \{\Psi^{i}_{r}\}^{T} \mathbf{F}_{s}
\]

The reduced coupling matrix is defined by:

\[
\mathbf{C}_{r} = \{\Psi^{i}_{r}\}^{T} \mathbf{C}_{s}
\]

and the reduced displacements are defined by:

\[
\mathbf{w}_{r} = \{\Psi^{i}_{r}\}^{T} \overline{\mathbf{w}}
\]

Reduced on the modal basis, the discretized structural equation Eq. 9 in terms of reduced displacements and jump of acoustic pressure becomes:

\[
[\mathbf{K}_{r} - \omega^2 \mathbf{M}_{r}] \{\mathbf{w}_{r}\} = \{\mathbf{F}_{r}\} - \mathbf{C}_{r} \{\mathbf{p}\}
\]
whose dimension is $m \times m$, with $m$ the number of modes only instead of the physical displacements on every nodes of the mesh of the structure.

In the equation Eq. 15, the exterior fluid around the structure is not taken into account so has no effect on the mechanical behavior of the elastic structure. This is the case where the mass of the fluid embedding the structure is negligible compared with the mass of that latter. In aerospace industry for example, the payload made of very light composite panels is not strongly affected by the fluid around which is the air so the modal basis is generally computed using Eq. 15. On the other hand, for marine applications, it is not the same situation. The heavy fluid, the water sea for example for underwater noise, has a strong effect on the mechanical behavior of the structure which needs to be taken into account during the computational method to evaluate the modal basis. Several numerical methods [22, 23] have been developed to compute that added mass matrix and are available nowadays and implemented in commercial FE packages. Added mass may be modelled by acoustic finite elements, as implemented for example in FE software ABAQUS. In the present paper the boundary element method, implemented in the FE software NX Nastran [21], is considered. A brief overview of the well-known theory of the BEM for the computation of added mass matrix is given in [22] allowing direct computation of wetted natural frequencies by solving

$$\det(K_B - \omega^2(M_B - M_A)) = 0$$

(22)

This method will be applied to an academic case of elastic sphere immersed in water. Another way to take into account the strong coupling of the structure in the heavy fluid around consists in solving the direct nodal mechanical system Eq. 10 instead of representing the mechanical behavior though its modal basis. Indeed, for real industrial applications, the modal basis computation is very expensive in terms of CPU time and RAM. It is even more true for the wetted modal basis which is almost impossible for very large industrial problem like a full submarine as illustrated in Fig 1.

3 THE HYBRID FINITE/BOUNDARY ELEMENT METHOD FOR THE ACOUSTIC DOMAIN

Concerning the propagation of the acoustic wave in the unbounded domain, the standard BEM approach or the more recent coupled MLMM as in [ESA ESTEC] can be employed. Nevertheless, even if this method has several positive points, it also has some important weaknesses limiting its use. In this part, we will present the hybrid approach which seems to be more appropriate.

(a)Vertical cut of the mesh of the submarine for the hybrid method (b)Positions of the fictitious surface embedding the obstacle

Figure 3: The fictitious surface for the Hybrid Method.
As illustrated in Fig. 3, the domain of computation $\Omega_f$ is truncated by introducing a fictitious surface $\Gamma$ embedding the initial obstacle like the Infinite Element Method or the FEM coupled with Perfectly Matched Layers. But contrary to the latters, that fictitious surface can be located at any distance $r$ of the obstacle and can take any shape. In the figure Fig. 3, the surface $\Gamma$ is a simple extrusion of the initial surface $\Sigma_f$ of the obstacle along the normal to the surface at $r = \lambda/20$ at 1kHz for example.

By using the integral formulation, we represent the solution at every point in a homogeneous domain by the data of the unknowns on the surfaces on this domain only by using one of the standard three-dimensional integral formulas. Those representations are valid for an exterior acoustic medium $\Omega_f$ with a smooth surface $\Sigma_f$ taking the following forms [5]:

$$\phi(x) = \phi(x) + V^{+}p_n(x) - N^{+}p(x), \quad x \in \Omega_f$$  \hspace{1cm} (23)

where

$$V^{+}p_n(x) = \int_{\Sigma_f} G_k(x, y) \partial_n p(y) d\Sigma_f(y)$$  \hspace{1cm} (24)

and

$$N^{+}p(x) = \int_{\Sigma_f} \partial_n G_k(x, y) p(y) d\Sigma_f(y)$$  \hspace{1cm} (25)

or

$$\partial_n \phi(x) = \partial_n \phi(x) + \partial_n V^{+}p_n(x) - \partial_n N^{+}p(x), \quad x \in \Omega_f$$  \hspace{1cm} (26)

where

$$\partial_n N^{+}p(x) = D^{+}p(x)$$  \hspace{1cm} (27)

The free-space Green's function $G_k$ for the Helmholtz equation in the three dimensions is given by

$$G_k(x, y) = e^{ik\|x-y\|}/4\pi\|x-y\|$$  \hspace{1cm} (28)

where $r = \|x - y\|$ is the distance between the field point $x$ and the moving point $y$ and $n$ is the outward directed normal at $y$. By using the integral representations Eq. 23 and Eq. 26, the radiation boundary condition on $\Gamma$ takes the following form:

$$\frac{\partial p}{\partial n} + ikp = \frac{\partial \phi}{\partial n} + ik\phi \text{ on } \Gamma$$  \hspace{1cm} (29)

with the normal vector $n$ on $\Gamma$ and $\Sigma_f$ pointing inside the domain $\Omega_{FEM}$ defined by $\Omega_{FEM} \subset \Omega_f$ with $\partial \Omega_{FEM} = \Gamma \cup \Sigma_f$. The acoustic equations in the initial system Eq. 9 becomes
The Sommerfeld radiation boundary condition is replaced by the radiation boundary condition Eq. 29 on $\Gamma$. Applying the standard Finite Element Method by multiplying the Helmholtz system Eq. 25 by a test function $p'$, the variational formulation of the system Eq. 30 and the Eq. 8 provides the following form

$$\forall p' \in H^1(\Omega_{FEM}), \text{ find } p \in H^1(\Omega_{FEM})/$$

$$H_{d_{FEM}}(p, p') - \omega^2 Q_{d_{FEM}}(p, p') - i\omega B_f(p, p') +$$

$$\omega^2 \partial_n V_f^r(p, p') - D_f^r(p, p') + i\omega \frac{2}{\varepsilon} \left( \omega^2 V_f^r(p, p') - N_f^r(p, p') \right) +$$

$$\omega^2 C_f(p, p') = -\frac{1}{\rho_f} \left( C_f(\partial_n \Phi_f, p) + i\omega \frac{2}{\varepsilon} C_f(\Phi_f, p') \right)$$

with $w = \overline{w} \overline{n} = \frac{1}{\rho_f \omega^2} \frac{\partial p}{\partial n}$ and where

$$H_{d_{FEM}}(p, p') = \int_{\Omega_{FEM}} \frac{1}{\rho_f} \nabla p \cdot \nabla p' \ d\Omega \quad ; \quad Q_{d_{FEM}}(p, p') = \int_{\Omega_{FEM}} \frac{1}{\rho_f} p \ p' \ d\Omega$$

$$B_f(p, p') = \int_{\Sigma_f} \frac{n}{\rho_f} \ p \ p' \ d\Sigma \quad ; \quad C_f(g, p') = \int_{\Sigma_f} \ n \ g \ p' \ d\Sigma$$

and with

$$V_f^r(p, p') = \int_{\Gamma_f} \overline{w}(y) G_k(x, y) p'(x) \ d\Gamma_f(y)$$

$$\partial_n V_f^r(p, p') = \int_{\Gamma_f} \overline{w}(y) \partial_n G_k(x, y) p'(x) \ d\Gamma_f(y)$$

$$D_f^r(p, p') = \int_{\Gamma_f} \frac{1}{\rho_f} p(y) \frac{\partial^2 G_k(x, y)}{\partial n^2} p'(x) \ d\Gamma_f(y)$$

and

$$N_f^r(p, p') = \int_{\Gamma_f} \frac{1}{\rho_f} p(y) \partial_{n_y} G_k(x, y) p'(x) \ d\Gamma_f(y) \ d\Gamma_f(x)$$

Discretizing the Eq. 31 by using the volumic Finite Elements in $\Omega_{FEM}$ and the surfacic Finite Elements on $\Sigma_f$ and $\Gamma$ and combining Eq. 10 and Eq. 31, finally the fully coupled vibro-acoustic fluid structure system expressed in terms of accelleration defined by $\ddot{\mathbf{y}} = -\omega^2 \mathbf{w}$ for the structure part is given by

$$\begin{bmatrix}
-\frac{1}{\omega^4} \left[ K_{II} - \omega^2 M_{II} \right] & C_{II} & 0 & 0 \\
C_{II} & -\left[ H_{d_{d_{II}}} - \omega^2 Q_{d_{d_{II}}} \right] & 0 & 0 \\
\partial_r V_f^r + i\omega \frac{2}{\varepsilon} V_f^r & \partial_r^2 V_f^r + i\omega \frac{2}{\varepsilon} N_f^r & -\left[ H_{d_{d_{II}}} - \omega^2 Q_{d_{d_{II}}} \right] & 0 \\
0 & -\left[ H_{d_{d_{II}}} - \omega^2 Q_{d_{d_{II}}} \right] & -\left[ H_{d_{d_{II}}} - \omega^2 Q_{d_{d_{II}}} \right] & -\left[ H_{d_{d_{II}}} - \omega^2 Q_{d_{d_{II}}} \right]
\end{bmatrix} \begin{bmatrix}
\ddot{\mathbf{y}}_p \\
\mathbf{p}_f \\
\mathbf{F}_t \\
\mathbf{p}_{\text{act}}
\end{bmatrix} = \begin{bmatrix}
\frac{1}{\rho_f} (\partial_n \Phi_f) + i\omega \frac{2}{\varepsilon} (\Phi_f) \\
0
\end{bmatrix}$$
where \( \{p_{\Sigma_f}, p_{\Gamma}, p_{\Omega_{FEM}}\} \) represents respectively the nodal values of the computed pressure on the surface \( \Sigma_f \) of the obstacle, on the fictitious \( \Gamma \) surface and in the interior of the fluid domain \( \Omega_{FEM} = \Omega_{FEM} \setminus \{\Sigma_f \cup \Gamma\} \) and \( \{\vec{y}\} \) is the nodal values vector of the computed acceleration of the elastic structure.

The linear system Eq. 35 can be solved by using a direct approach but several difficulties limit this way:

- The left-hand-side (LHS) is composed by both linear sparse and dense operators and mixt operators are not easy to store and to inverse. The only way consists in the Schur Complement of the Sparse part over the dense one.
- The global linear system is not symmetric.
- The dense blocks are treated by standard BEM method whose RAM and CPU time requirements increase as \( O(N^2) \) and \( O(N^3) \) to be stored and inversed respectively with \( N \) standing for the number of degrees of freedom over the surface \( \Sigma_f \) of the obstacle.

The best solution to treat the linear system Eq. 35 relies on a Domain Decomposition Method (DDM) with overlapping ordering the sparse and the dense operators to obtain the following form:

\[
\begin{bmatrix}
-\frac{1}{\omega^2}[(K_{\Omega} - \omega^2 M_{\Omega})]_{\Sigma_f} & C_{\Sigma_f} & 0 & 0 \\
C_{\Sigma_f} & -[(H_{\Omega_{FEM}} - \omega^2 Q_{\Omega_{FEM}})]_{\Sigma_f,\Sigma_f} & 0 & -[(H_{\Omega_{FEM}} - \omega^2 Q_{\Omega_{FEM}})]_{\Sigma_f,\Gamma} \\
0 & 0 & -[(H_{\Omega_{FEM}} - \omega^2 Q_{\Omega_{FEM}})]_{\Gamma,\Sigma_f} & -[(H_{\Omega_{FEM}} - \omega^2 Q_{\Omega_{FEM}})]_{\Gamma,\Gamma} \\
0 & 0 & -[(H_{\Omega_{FEM}} - \omega^2 Q_{\Omega_{FEM}})]_{\Gamma,\Gamma} & -[(H_{\Omega_{FEM}} - \omega^2 Q_{\Omega_{FEM}})]_{\Gamma,\Omega_{FEM}} \\
\end{bmatrix}
\begin{bmatrix}
\vec{y} \\
\vec{p}_{\Sigma_f} \\
\vec{p}_{\Gamma} \\
\vec{p}_{\Omega_{FEM}}
\end{bmatrix} = G(F_{\Sigma_f}, \phi_\psi, \partial_n \phi_\psi, p_{\Sigma_f}, \partial_n p_{\Sigma_f})
\]

where

\[
G(F_{\Sigma_f}, \phi_\psi, \partial_n \phi_\psi, p_{\Sigma_f}, \partial_n p_{\Sigma_f}) = \begin{bmatrix}
F_{\Sigma_f} \\
0 \\
\begin{bmatrix}
1 & (C_{\Sigma_f})[(\partial_n \phi_\psi) + i\omega \frac{\eta}{c} [C_{\Sigma_f}] (\phi_\psi)] - [p_{\Sigma_f} \vec{x}_f] + i\omega \frac{\eta}{c} N[f, \vec{x}_f] \{p_{\Sigma_f}\} - [\partial_n V_{\vec{x}_f}] + i\omega \frac{\eta}{c} V_{\vec{x}_f} \{\vec{y}\}
0
\end{bmatrix}
\end{bmatrix}
\]

On that form, noting that \( [C_{\Sigma_f}]^t = [C_{\Sigma_f}] = [C] \), the linear system becomes symmetric and completely sparse. In the right-hand-side (RHS), both quantities \( \{p_{\Sigma_f} = p_{\Omega_{FEM}}\} \) and \( \{y_{\Sigma_f} = \vec{y}\} \) are still unknown so the system needs to be solved implicitly by an iterative process. To solve the problem Eq 36 under the saddle point form \( AX^{(n+1)} = F - BX^{(n)} \), the standard Gauss-Seidel method can be used. It consists in the following algorithm.
\[
\begin{align*}
\text{Define: } & \varepsilon, \text{Nb}_{\text{iter}}^{\text{max}} \\
\text{Initiate: } & n = 0, p_{\Sigma_f}^{(0)}, \gamma^{(0)}, p_{\Sigma_f}^{(1)} \neq 0 \text{ and } \gamma^{(1)} \neq 0 \\
\text{While } & \| R(\{p_{\Sigma_f}^{(n+1)}, \gamma^{(n+1)}\}, \{p_{\Sigma_f}^{(n)}, \gamma^{(n)}\}) \| > \varepsilon \text{ and } n < \text{Nb}_{\text{iter}}^{\text{max}} \\
& (1) \text{ Compute } G_n(F_{\Sigma_f}, \Phi, \partial_n \Phi, p_{\Sigma_f}^{(n)}, \gamma^{(n)}) \text{ by MVP} \\
& (2) \text{ Solve the linear system for } \{p_{\Sigma_f}^{(n+1)}\} \text{ and } \{\gamma^{(n+1)}\} \\
& (3) \text{ Compute the new residual } R(\{p_{\Sigma_f}^{(n+1)}, \gamma^{(n+1)}\}, \{p_{\Sigma_f}^{(n)}, \gamma^{(n)}\}) \\
& (4) n = n + 1 \\
\end{align*}
\]

where \( R(p, q) = \| p - q \| / \| q \| \) is the residual vector whose norm measures the error of the resolution during the iterative solver. Before starting the resolution of Eq. 36, the FEM LHS, which is completely sparse, is stored/computed and a LU factorisation is done. During the iterative process, at step (1), some matrix-vector products (MVP) are needed to evaluate the new iterate \( G_{n+1} \). This step is handled by the MLFMM algorithm; this operation is computed very quickly and with a very low RAM requirement. At step(2), the resolution of the linear system consists only of front/backward substitution, the linear system being already factorized at the beginning of the process. Mathematically, it has been demonstrated in [11], [12], [13] the problem Eq. 36 is well-posed and its numerical solution converges towards the solution of the initial problem Eq. 9. It is very interesting to note the first iteration of the process Eq. 38 (with \( p_{\Sigma_f}^{(0)} = 0 \text{ and } \gamma^{(0)} = 0 \)) is exactly the FEM formulation with homogenous impedance boundary condition. During the iterative process, the radiation boundary condition will be corrected until it reaches the convergence criteria, that is the reason why this hybrid method is called adaptive absorbing boundary condition (AABC). Consequently, the further from the obstacle the fictitious surface \( \Gamma \) is, the faster the iterative solver will converge (\text{Nb}_{\text{iter}}^{\text{HYB}} \text{ will be very small}) but the larger the domain of computation \( \Omega_{\text{FEM}} \) will be. Depending on the computer resources and also the mesh effort the user would like to spend to model the problem, there exists a compromise between both criteria. It has been also demonstrated numerically in [11], [12], [13] the convergence of the iterative process is very fast (\text{Nb}_{\text{iter}}^{\text{HYB}} < 20 \text{ iterations}) even when the surface \( \Gamma \) is very close to the obstacle (\( d < \lambda/100 \)). This approach presents all the strengths of the other methods and no weakness:

- The first and main interest of this method is it leads to sparse matrix operators, i.e not fully populated; the RAM requirement to store those sparse operators is \( O(N) \) and the complexity to build them is \( O(N) \) where \( N \) stands for the number of nodes of the mesh descretizing the volumic domain of computation \( \Omega_{\text{FEM}} \) which can be chosen very small.
- The final equation Eq. 36, which is symmetric, is solved by using the MUMPS library based on a multi-frontal approach which performs a direct \( LDL^T \) factorization [4] whose complexity is \( O(N^2) \). The direct approach is very well adapted to multiple load cases, i.e. multiple plane waves to compute monostatic TS with a very large number of angles.
- For this method, the domain of computation is quasi-minimal. The fictitious surface \( \Gamma \) can be located at any distance from the obstacle and takes any shape.
- The radiation boundary condition is exact on the surface \( \Gamma \) at the convergence of the iterative process.
The integral operators are never fully allocated and their inversion is not needed. Only some MVP are computed by MLFMM during the iterative process of resolution. Thanks to this method, the RAM requirement to store all integral operators is on $O(N)$ and the complexity to update the (RHS) is $N \log_{10}(N)$ only.

Finally, last asset and not the least, the integral operators are no longer singular because the surfaces $\Gamma$ and $\Sigma_f$ are not connected.

To accelerate the convergence of the iterative process, it can be more appropriate to use a Krylov approach rather than a standard Gauss-Seidel method. We design by $\Lambda$ the saddle point operator in Eq. 36 defined by:

$$X = \Lambda(X, B)$$

Consequently, the residual between two iterates is given by

$$R = \Lambda(X, B) - X$$

Because the operator $\Lambda$ is bi-linear in $X$ and $B$, we can write the system in Eq 39 under the following form

$$X - \Lambda(X, 0) = \Lambda(0, B)$$

To solve this final system using a Krylov method like the GMRES algorithm, the RHS ($\Lambda(0, B)$) is given by one iteration of the Gauss-Seidel process with initial condition set to zero taken into account the effect of the sources. The MVP ($X - \Lambda(X, 0)$) is given by the difference between the current data and result of one iteration of the Gauss-Seidel process by fixing the sources to zero. It is interesting to note that in the linear system Eq 36, the structure can be represented under modal or nodal unknowns depending on the capabilities to compute the modal basis taking into account the added mass effect. When the frequency increases, the modal approach is no more useable and the nodal method is the only solution to model the problem. In this case, both elastic structure and acoustic fluid domain represented by physical unknowns are perfectly managed by a Distributed Memory Processing dedicated sparse linear system like the MUMPS library for example which is able to deal with very large number of unknowns. In the last part, some applications for underwater TS computation for large elastic target are presented to demonstrate the capabilities of that hybrid approach.

4 APPLICATIONS

In this final section, we will present some numerical experiments comparing different approaches and demonstrating the strengths of the coupled hybrid FEM/MLFMM AABC approach.

4.1 Rigid application: the BETSSI-sub model

Since TS data on real submarines is kept confidential, a benchmark model has been designed being realistic enough to provide an accurate measure of real-world performance. It has been developed at Forschungsanstalt der Bundeswehr für Wasserschall und Geophysik (FWG) in Kiel, Germany in 2002 [14]. The BeTSSi-Sub test case, for Benchmark Target Strength Simulation Submarine, is a 62m x 11m x 7m submarine as illustrated in Fig. 4. For this
simulation, the submarine is considered as rigid. The size of the meshes are presented in Table 1 at 200Hz and 1kHz and in table 4 at 4kHz. All the numerical simulations have been done by using the vibro-acoustic VA One Software [15] developed by ESI Group.

Figure 4: The BeTSSi-Sub test case/The BEM/FMM mesh of the BETSSI model @1kHz.

For each method, the surface $\Sigma_f$ of the obstacle contains 82,263 nodes (164,522 elements). The original shape as well as the details of the submarine are very well taken into account by the mesh whose size is fixed to $\lambda/10$ at 1kHz. The BEM and the MLFMM solvers use exactly the same mesh: only the surface $\Sigma_f$ of the obstacle. The surface $\Gamma$ is only an extrusion of the surface $\Sigma_f$ located at $d = \lambda/20$ at 1kHz for the AABC hybrid FEM-BEM solver. The domain $\Omega_{FEM}$ for the AABC hybrid FEM-BEM solver contains only 164,807 nodes (495,022 elements). From the meshing point of view, this AABC hybrid FEM-BEM method enables a considerable reduction of the domain of computation $\Omega_{FEM}$ in comparison to the mesh which would be needed for the standard FEM coupled with PML or IEM approaches. The same meshes are used for both frequencies: 200Hz and 1kHz.

Table 1: Size of the mesh for each method used at 200Hz and 1kHz.

<table>
<thead>
<tr>
<th>Methods</th>
<th>Nodes on $\Sigma_f$</th>
<th>Nodes on $\Gamma$</th>
<th>Nodes in $\Omega_{FEM}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>BEM/MLFMM</td>
<td>82,263</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>AABC Hybrid</td>
<td>82,263</td>
<td>82,263</td>
<td>164,807</td>
</tr>
</tbody>
</table>

4.1.1 Bistatic TS

Figures Fig. 4 and Fig. 5 compare the BEM total surface pressure as a reference to the one obtained by the hybrid FEM-BEM for a unit strength plane wave source at a broadside angle ($\alpha_i = 90^\circ, \beta_i = 90^\circ$) and at two frequencies of 200Hz and 1kHz. The results show perfect agreement between both methods. The relative errors of the total pressure field on the surface $S$ computed by the AABC hybrid FEM-BEM method is lower than 1% for both frequencies with the BEM results considered as a reference. The complexity and the RAM requirements for the different methods are presented in table 2 for 200Hz and 1kHz. The CPU Time and the RAM requirements for the different methods are presented in table 3 for 4kHz for the bistatic TS plane wave source, broadside incidence. For the bistatic TS, it is easy to see the best candidates are the MLFMM and Hybrid FEM/MLFMM AABC in terms of CPU time but also of RAM requirements. For very complex physics (like cavity for example), for very large number of nodes $N$ (so large obstacle and/or high frequencies), the number of iterations needed by the MLFMM solver $N_{iter}^{FMM}$ can be very large (several hundreds) but the number of iteration needed by the Hybrid FEM/MLFMM AABC solver $N_{iter}^{HYB}$ stays low (less than 20).
Figure 4: Total acoustic field for the BeTSSi submarine for the BEM (left) and AABC hybrid FEM-BEM (right) from a 200Hz plane wave source, broadside incidence.

Figure 5: Total acoustic field for the BeTSSi submarine for the BEM (left) and AABC hybrid FEM-BEM (right) from a 1kHz plane wave source, broadside incidence.

Table 2: Complexity and RAM requirements for each method.

<table>
<thead>
<tr>
<th>Methods</th>
<th>Complexity</th>
<th>RAM (Gb)</th>
</tr>
</thead>
<tbody>
<tr>
<td>BEM</td>
<td>$O(N^3)$</td>
<td>$O(N^2)$</td>
</tr>
<tr>
<td>MLFMM</td>
<td>$O(N_{FMM}^{iter} \times N \log^{2}<em>{10}(N) \times N</em>{RHS})$</td>
<td>$O(N \log_{10}^{2}(N))$</td>
</tr>
<tr>
<td>AABC Hybrid</td>
<td>$O(N_{HYP}^{iter} \times N \log^{2}<em>{10}(N) \times N</em>{RHS})$</td>
<td>$O(N \log_{10}^{2}(N))$</td>
</tr>
</tbody>
</table>

Figures Fig. 6 and Fig. 7 compare both collocation and variational MLFMM total surface pressures as a reference to the one obtained by the hybrid FEM-BEM for a unit strength plane wave source at a broadside angle ($\alpha_i = 90^\circ, \beta_i = 90^\circ$) at the frequency of 4kHz. The collocation MLFMM is used in SMP version with 16 processors whereas the variational MLFMM and hybrid AABC are used in sequential version (with 1 processor only). It is shown in table 3 the hybrid AABC needs the lowest level of RAM (16 Gb only) against both versions of MLFMM (21 and 40 Gb). Concerning the CPU Time, the hybrid AABC method reaches the convergence in only 9 iterations (3h49mn) whereas both MLFMM versions need greater CPU Time (36h15mn in SMP16 for the collocation MLFMM and 78h42mn in sequential for the variational MLFMM). The results show perfect agreement between all the methods. The relative errors of the total pressure field on the surface $S$ computed by the AABC hybrid FEM-BEM method is lower than 1% with the variational MLFMM result considered as a reference.
Figure 6: Total acoustic field for the BeTSSi submarine for the AABC hybrid FEM-BEM (top left), Collocation MLFMM (top right) and Variational MLFMM (down) from a 4kHz plane wave source, broadside incidence.

Figure 7: Total acoustic field for the BeTSSi submarine for the AABC hybrid FEM-BEM (top left), Collocation MLFMM (top right) and Variational MLFMM (down) from a 4kHz plane wave source, broadside incidence.

Table 3: CPU Time and RAM requirements for each method at 4kHz.

<table>
<thead>
<tr>
<th>Methods</th>
<th>CPU Time</th>
<th>RAM (Gb)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Collocation MLFMM</td>
<td>36h15mn (459 iterations)</td>
<td>40</td>
</tr>
<tr>
<td>Variational MLFMM</td>
<td>78h42mn (246 iterations)</td>
<td>21</td>
</tr>
<tr>
<td>AABC Hybrid</td>
<td>3h49mn (9 iterations)</td>
<td>16</td>
</tr>
</tbody>
</table>

Table 4: Size of the mesh for each method used at 4kHz.

<table>
<thead>
<tr>
<th>Methods</th>
<th>Nodes on $\Sigma_f$</th>
<th>Nodes on $\Gamma$</th>
<th>Nodes in $\Omega_{FEM}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>MLFMM</td>
<td>233,329</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>AABC Hybrid</td>
<td>233,329</td>
<td>233,329</td>
<td>467,999</td>
</tr>
</tbody>
</table>

4.1.2 Monostatic TS

Figures Fig. 8 and Fig. 9 show the monostatic TS as a function of backscattered angle for the BeTSSi submarine, as calculated by the acoustic BEM and AABC methods (at 2° increments) for 200Hz and 1kHz which represent respectively a
\( kl = 52 \) and \( kl = 260 \). RAYON stands for the deterministic BEM/FEM/AABC solvers inside the VA One software. Also included are the TS results for the BeTSSI submarine model from AVAST software (Acoustic Vibration And STructural analysis) developed by Martec Ltd., Halifax, Canada [16]. The TS results show very good agreement for 200Hz for all the aspect angle range between AVAST, BEM and AABC. Both BEM and AABC give exactly the same results for 200Hz and 1kHz. Some small differences can be noticed on the second half of the aspect angle range \( 120^\circ < \alpha < 180^\circ \) for 1kHz. The meshes used for BEM/AABC on one hand and AVAST on the other hand are not the same. In that aspect angle range, the results of the TS are very sensitive to the mesh because of the scattering by the shape of the obstacle (the rudders and the hydroplane blades). The results can be also improved by reducing the aspect angle increments (a \( 2^\circ \) increment is used for BEM and AABC and probably lower for AVAST).

Figure 8: Total acoustic field for the BeTSSI submarine for the BEM (left) and AABC hybrid FEM-BEM (right) from a 200Hz plane wave source, broadside incidence.

Figure 9: Monostatic target strength as a function of aspect angle for the BeTSSI submarine at a frequency of 1kHz for BEM and AABC method.

Fig. 10 shows finally the monostatic TS as a function of backscattered angle for the BeTSSI submarine, as calculated by the acoustic AABC method (at \( 2^\circ \) increment) for 4kHz which
represents a $kL = 1040$. Also included are the TS results for the BeTSSi submarine experimental measurements from DSTO (Defense Science and Technology Organisation) Australia [17].

Comparatively, the AABC Hybrid FEM-BEM method solves the monostatic problem (which is that hardest problem because of the multiple plane waves) about 8 times faster than the BEM using approximately 12 times less memory for both frequencies. The iterative solver Eq. 38 reaches the convergence criteria in only 8 iterations for 200Hz and 14 iterations for 1kHz for $r$ located at $\lambda/20$ at 1kHz. The MLFMM is comparable to the AABC Hybrid FEM-BEM method for a single plane wave for 200Hz but is the slowest one to deal with the 91 plane waves and for 1kHz for the number of iterations to reach the convergence. The reduced computational and memory requirements of the AABC Hybrid FEM-BEM allow large problems with many unknowns to be solved on desktop PCs. The FEM, the MLFMM and the AABC Hybrid FEM-BEM method run on an i7 2.7GHz processor with 16Gb of RAM on a single processor desktop PC and the BEM method needs to be run on a 2.7GHz processor with 128Gb of RAM on a multiple processors linux machine.

### 4.2 Fully coupled application: the shell elastic sphere immersed in water

In this example, we will investigate the sound scattered by an elastic spherical shell impinged by an acoustic plane wave. An analytical solution presented in Junger and Feit [27] or [28] is used to validate the accuracy of the results obtained with the presented method.
An elastic spherical shell with middle surface radius $R = 1 \text{ m}$, thickness $h = 0.01 \text{ m}$, and centered at the origin $(0, 0, 0)$ is considered. A plane wave of amplitude $p_0 = 1$ propagating in the $+z$ direction is scattered by the sphere. The fluid medium surrounding the sphere is assumed to be water with a sound speed of $c = 1500 \text{ m.s}^{-1}$ and an ambient density of $\rho_f = 1000 \text{ Kg/m}^3$. The spherical shell is assumed to be made of steel with young’s modulus $E = 195\text{ GPa}$, density $\rho_s = 7700 \text{ Kg/m}^3$ and poisson’s ratio $\nu = 0.29$. We are interested in computing the sound scattered by the elastic sphere at $10\text{ m}$.

### 4.2.1 Finite Element Analysis

We performed a modal analysis on unconstrained sphere using the MFLUID card in the NASTRAN finite element software to estimate invacuo modes and natural frequencies. The natural frequencies for the above steel shell in a vacuum and submerged in water, are tabulated in Table 5 and compared between the theory presented in [28] and the numerical results.

The first few natural frequencies along with their degeneracy in parenthesis are $0 \text{ Hz} (6)$, $590.11 \text{ Hz} (5)$, $698.53 \text{ Hz} (7)$, $741.85 \text{ Hz} (9)$, etc. The first six modes with zero frequencies belong to the rigid body motions of the sphere. It can be noticed the results are coherent with the theory even in the submerged case.

<table>
<thead>
<tr>
<th>Mode</th>
<th>In vacuo (theory)</th>
<th>Submerged (theory)</th>
<th>In vacuo (numeric)</th>
<th>Submerged (numeric)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
<td>1E-5 (6)</td>
<td>1E-5 (6)</td>
</tr>
<tr>
<td>2</td>
<td>589.66</td>
<td>272.15</td>
<td>590.11 (5)</td>
<td>268.59 (5)</td>
</tr>
<tr>
<td>3</td>
<td>697.09</td>
<td>343.77</td>
<td>698.53 (7)</td>
<td>347.76 (7)</td>
</tr>
<tr>
<td>4</td>
<td>740.07</td>
<td>391.52</td>
<td>741.85 (9)</td>
<td>395.76 (9)</td>
</tr>
<tr>
<td>5</td>
<td>761.55</td>
<td>429.71</td>
<td>764.64 (11)</td>
<td>435.07 (11)</td>
</tr>
</tbody>
</table>
In total until 1165Hz, 368 modes have been computed for the in vacuo case against 540 modes for the submerged case. It may be seen that radiation loading reduces the natural frequencies of the shell therefore the number of modes representing the mechanical behavior of the structure is considerably increased. To compute the modal basis in vacuo, the global CPU time is 7mn against 1h34mn for the submerged case with 20 CPUs because of the initial computation of the Added Mass Matrix in BEM (outofcore in Nastran) and the impact of that fully populated matrix Added Mass Matrix in the eigen modes algorithm. In terms of CPU time, the wetted modal Hybrid AABC needs 28mn and 380Mb of RAM whereas the nodal Hybrid AABC needs 34mn and 851Mb with a sequential version.

<table>
<thead>
<tr>
<th></th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>775.88</td>
<td>458.36</td>
<td>779.77 (13)</td>
<td>463.91 (13)</td>
</tr>
<tr>
<td></td>
<td>790.20</td>
<td>487.01</td>
<td>792.32 (15)</td>
<td>490.83 (15)</td>
</tr>
<tr>
<td></td>
<td>802.14</td>
<td>510.88</td>
<td>804.90 (17)</td>
<td>513.93 (17)</td>
</tr>
<tr>
<td></td>
<td>814.07</td>
<td>534.76</td>
<td>819.32 (19)</td>
<td>537.01 (19)</td>
</tr>
</tbody>
</table>

Fig. 12 shows the back-scattering at 10m (Near TS) as a function of the frequency for the elastic spherical shell. It can be noted both nodal and wetted modal approaches give very coherent results against analytical results. For very large industrial applications when the computation of the wetted modal basis is really expensive, the nodal approach is the only solution to take into account the heavy fluid around the target.

12 CONCLUSIONS

As described, many methods for accurate modeling of the Target Strength of arbitrary shaped objects like a complex submarine are available but they all have some strong limitations coming from computational resources (RAM or CPU time). The AABC hybrid FEM-BEM approach presented in this paper tackles this problem. As shown, this method combines all the strengths of the other methods and presents no weakness. The distance between the fictitious surface supporting the radiation boundary condition and the obstacle is not restricted by the wavelength but rather by the computational resources available. Practically, this distance is fixed equal to...
the size of the mesh. By this way, the domain of computation is quasi-minimal. During the iterative process of resolution, only a sparse linear system is solved and the integral operators are involved only through matrix vector products very well adapted to the MLFMM algorithm. Further improvements to the current work, chiefly, the incorporation of high performance computing, will substantially increase its performance reducing the time of computation but also enabling higher frequencies to be reached, up to 8kHz or 10kHz for the rigid BETSI submarine. As presented, the current version of the software is also able to deal with coupled fluid-structure problems and has been applied on a simple academic case with both wetted modal and nodal approaches for the structure part. Nevertheless, it would be interesting to validate it against a more complex industrial case like a real submarine for example. This prototype is also able to deal with a rigid infinite plane to model the ground or an infinite pressure release surface to model the surface of the sea. It would be also interested to model a realistic model of sediments through a rigid infinite plane with an impedance to validate it.

REFERENCES
HYDRODYNAMIC ANALYSIS OF A SEMI-SUBMERSIBLE FLOATING WIND TURBINE. NUMERICAL VALIDATION OF A SECOND ORDER COUPLED ANALYSIS

Gutiérrez-Romero JE*, Serván-Camas B† and García-Espinosa J†.

*Escuela Técnica Superior de Ingeniería Naval y Oceánica (ETSINO)
Universidad Politécnica de Cartagena
Paseo Alfonso XIII, n 52, 30203
e-mail: jose.gutierrez@upct.es, web page: http://www.upct.es

† International Center for Numerical Methods in Engineering (CIMNE)
Universidad Politécnica de Cataluña
Edificio C1, Campus Norte UPC
Gran Capitán s/n, 08034 Barcelona, Spain
e-mail: bservan@cimne.upc.edu - Web page: http://www.cimne.com

Key words: Seakeeping, Second-order analysis, semisubmersible platform

Abstract. This work presents a comprehensive computational hydrodynamic analysis of a semisubmersible offshore wind turbine. The studied case is based on a three floater wind turbine designed for a 1.5 MW turbine. The analysis is supported by experimental tests and it is focused on the second-order response of the platform, including mooring effects. The different analysis carried out include decay test, regular waves and bichromatic waves. Numerical simulations are performed in time-domain using the computational model implemented in the SeaFEM solver.

The analysis of the obtained results includes the comparison of the numerical and experimental response of the platform. Special attention is paid to the second-order effects in the dynamic response of the system.

1 INTRODUCTION

There is a growing focus of the industry on Floating Offshore Wind Turbines (FOWT) for due to their ability to access the enormous wind resources available over deep water. Despite the existence of real scale prototypes already operating, such as the Hywind in Norway [1] or the Windfloat in Portugal [2], the industry still faces design and operation challenges which require the complete full coupled analysis of them. In this work, we present a full hydrodynamic coupled analysis for the HiPRWind semisubmersible [3] floating wind turbine model using up-to-second-order wave model coupled with non-linear finite element model for mooring analysis, which is validated with experimental tests carry out at ECN Nantes facilities.

The hydrodynamics of the semisubmersible concept for FOWTs has received some attention in the recent literature. For instance, [4] and [5] focused on slow-drift and mean-drift forces
of semisubmersible platforms, a comparison of a semisubmersible against a SPAR concept can be found in [6] and Simulations in the Time-Domain (TD) considering different models for the hydrodynamic loads were carried out in [7].

One of the main concerns regarding semisubmersible platforms are the slow-drift forces. These forces are usually in the range of the surge natural period of semisubmersible platform with catenary mooring lines, leading to large displacements when excited near the resonance frequency. And although the wave frequencies are usually larger than the natural periods, second-order effects contain low frequencies components that will probably excite slow-drift in the system platform-mooring. Therefore, large excursion might happen due to second-order effects.

Second-order forces might increase the surge response of semisubmersible platforms, even becoming larger that the first-order response. And although Neumann’s approximation could be used for estimating the slow-drift forces (which only depends on the first-order solution), it might not be enough precise as shown in [5]. Hence, second-order effect must be taken into account to accurately compute slow-drift motions, and design the mooring system accordingly. The impact of the slow-drift forces on the design of the mooring systems and the difficulties to estimate the corresponding forces is yet a problem that requires substantial research. Previous works as López-Pavón et al. [5] putted the focus on the estimation and verification of the second-order wave induced forces on the HiPRWind semisubmersible platform.

So, in this work, a Finite Element Method (FEM) that solves up to second-order in the time-domain model coupled with non-linear forces arising from the mooring lines are used to carry out a hydrodynamic analysis of semisubmersible HiPRWind platform [3, 5]. A verification of the model is carried out comparing to second-order analytical solutions available. The computer model of the HiPRWind platform is calibrated using decay tests and then analyzed in regular and bichromatic. Finally, some conclusions are made regarding the verifications and analysis presented.

2 PROBLEM STATEMENT

Then the problem statement about seakeeping of offshore structures is established. A brief descriptions about second-order and mooring model employed are established.

2.1 Second-order diffraction-radiation governing equations

The governing equations for the second-order diffraction-radiation wave problem are obtained applying Taylor expansion on the boundary surfaces of a time-independent domain. This approach allows to approximate the free surface on \( z = \zeta \) and the mean body surface \( \Gamma_0^b \) at time \( t \). Then, a perturbed solutions based on Stokes expansion procedure is applied to the velocity potential, free surface elevation, and body motion. More details can be found in [8, 9]. The solution can be decomposed as

\[
\varphi = \psi + \theta, \\
\xi = \eta + \zeta, 
\]

where \( \psi \) is the incident wave velocity potential, \( \theta \) is the diffraction-radiation wave velocity.
potential, $\eta$ is the incident wave elevation, and $\zeta$ is the diffraction-radiation wave elevation. Then, the wave diffraction-radiation governing equations up to second-order are [8, 9]:

$$\Delta \theta^{1+2} = 0 \quad \text{in } \Omega, $$

$$\frac{\partial \eta^{1+2}}{\partial t} - \frac{\partial \theta^{1+2}}{\partial z} = - \frac{\partial \theta^{1}}{\partial x} \frac{\partial \eta^{1}}{\partial x} - \frac{\partial \theta^{1}}{\partial y} \frac{\partial \eta^{1}}{\partial y}$$

on $z = 0$ (4)

$$\frac{\partial \eta^{1+2}}{\partial t} + P_{fs} \rho + g \eta^{1+2} = - \eta^{1} \frac{\partial}{\partial z} \left( \frac{\partial \theta^{1}}{\partial t} \right) - \zeta^{1} \frac{\partial}{\partial z} \left( \frac{\partial \theta^{1}}{\partial t} \right)$$

on $z = 0$, (5)

$$\mathbf{v}_{\theta}^{1+2} \cdot \mathbf{n}^{1} = - \left( \mathbf{v}_{\psi}^{1+2} + \mathbf{v}_{\psi}^{1+2} + \mathbf{r}_{b} \cdot \left( \nabla \mathbf{v}_{\theta} + \nabla \mathbf{v}_{\psi} \right) \right) \cdot \mathbf{n}^{1}$$

on $\Gamma_{b}$, (6)

$$\frac{P_{b}^{1+2}}{\rho} = - g z_{b} - g r_{b}^{1+2} - \frac{\partial \theta^{1+2}}{\partial t} - \mathbf{r}_{b} \cdot \nabla \left( \frac{\partial \theta^{1}}{\partial t} \right)$$

$$- \frac{1}{2} \nabla \theta^{1} \cdot \nabla \theta^{1} - \nabla \psi^{1} \cdot \nabla \theta^{1}$$

on $\Gamma_{b}$, (7)

where superscripts 1 and $1+2$ denote the components at the first-order and up to second-order solution, and $r_{b}$ is the displacement vector at a point over body.

2.2 Body Dynamics

The body dynamics of the floating body are governed by the equation of motion:

$$\ddot{\mathbf{M}} \mathbf{X}_{\text{rel}} + \ddot{\mathbf{K}} = \mathbf{F},$$

where $\ddot{\mathbf{M}}$ is the mass matrix of the body, $\ddot{\mathbf{M}}$ is the hydrostatic restoring coefficient matrix, $\mathbf{F}$ is the vector of the hydrodynamic forces induced by dynamic pressures plus any other external forces, and $\mathbf{X}$ represent the movements of the six degrees of freedom of the body.

Loads acting on the body are obtained by direct pressure integration on the body surface underneath the mean water level, except for the hydrostatic forces, which are obtained via the corresponding hydrostatic restoring matrices. Also, the second-order loads and moments ($\mathbf{F}_{wl}^{2}$ and $\mathbf{M}_{wl}^{2}$) due to the change of the wetted surface induced by the first order solution are accounted for:

$$\mathbf{F}_{wl}^{2} = - \frac{1}{2} \rho g \int_{\Gamma_{wl}} \left( \xi^{1} - r_{wp}^{1} \right)^{2} \mathbf{n}_{p}^{0} \sqrt{1 - (n_{wp}^{0})^{2}}$$

$$\mathbf{M}_{wl}^{2} = - \frac{1}{2} \rho g \int_{\Gamma_{wl}} \left( \xi^{1} - r_{wp}^{1} \right)^{2} \mathbf{G}^{0} \mathbf{P}^{0} \mathbf{n}_{p}^{0} \sqrt{1 - (n_{wp}^{0})^{2}}$$
where $\Gamma_0$ is the mean wetted surface, $\xi$ is the free surface elevation, $n_p$ is the initial body surface normal vector at point P located on wet surface, $G_0 P_0$ is the vector from the center of gravity of the floater G to any point P on the wet surface. Details of each component can be found in [8, 9].

2.3 Equations for cable dynamics

The dynamic equations for a mooring cable with length $L$ with negligible bending and torsional stiffness can be formulated as [10]

$$\left(\rho_w C_m A_0 + \rho_0\right) \frac{\partial^2 \mathbf{r}_l}{\partial t^2} = \frac{\partial}{\partial l} \left( E A_0 \frac{e}{e + 1} \frac{\partial \mathbf{r}_l}{\partial l} \right) + \mathbf{f}(t) \left(1 + e\right),$$

(11)

where $\rho_w$ is the water density, $C_m$ is the added mass coefficient, $\rho_0$ is the mass per unit length of the unstretched cable, $\mathbf{r}_l$ is the position vector, $E$ is the Young’s modulus, $A_0$ is the cross-sectional area of the cable, $e$ is the strain, $\mathbf{f}$ are the external loads applied on the cable, and $l$ is the length along the unstretched cable. The external loads acting on the cable, considered in this work, are the self-weight of the cable, hydrostatic loads, drag forces, and seabed interaction.

3 ANALYSIS OF HIPRWND FLOATING WIND TURBINE

3.1 Introduction

The floating platform geometry considered in this work has been provided by the HiPRWind FP7 project [3, 11] and is composed by three buoyant columns connected by bracings. Model tests were carried out at ECN Nantes’ facilities. A model built in stainless steel with scale $\lambda =1/19.8$ was used in the tests (see Figure 1). Table 1 provides the platform particulars in full scale, as well as the water depth considered for this study. The mesh model consists of tetrahedral 643603 elements and 119350 nodes. The Figure 2 shows a caption of the mesh used to carry out the hydrodynamic study.

Figure 1: HiPRWind platform model at wave basin.
Table 1: Main particulars of HiPRWind semisubmersible platform.

<table>
<thead>
<tr>
<th>Item</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Operation design draft</td>
<td>100.0 m</td>
</tr>
<tr>
<td>Depth</td>
<td>15.5 m</td>
</tr>
<tr>
<td>Distance between column centres</td>
<td>35.0 Kg</td>
</tr>
<tr>
<td>Column diameter</td>
<td>7.0 m</td>
</tr>
<tr>
<td>Heave plates diameter</td>
<td>20.0 m</td>
</tr>
<tr>
<td>Displacement</td>
<td>2332.0 Tons</td>
</tr>
<tr>
<td>Centre of gravity ((x_g; y_g; z_g))</td>
<td>(0.0; 0.0; -4.46) m</td>
</tr>
<tr>
<td>Number of mooring lines</td>
<td>3</td>
</tr>
<tr>
<td>Stiffness</td>
<td>634 MN</td>
</tr>
<tr>
<td>Length</td>
<td>330.0 m</td>
</tr>
<tr>
<td>Weight per unit length</td>
<td>1453.90 N/m</td>
</tr>
</tbody>
</table>

Figure 2: FEM Model and computational mesh overview.

3.2 Calibration of numerical model

In order to predict seakeeping in real conditions with a potential flow solver, viscous effects are to be incorporated via external forces. These external forces are simplified formulas accounting for the overall viscous effects acting on the platform. The viscous effects have been included in the computational solver by means of linear and quadratic damping models. This model has been calibrated using the experimental information of the extinction tests for surge, heave and pitch motions. Three elastic lines were used to keep the position of the model during the extinction experiments. These lines have a small linear weight distribution so the catenary effect is negligible. A pretension of 550 kN were applied to each line.

The viscous damping forces have been divided into two groups. The first group corresponds to the bracings of the structure. The corresponding forces are applied in the center of gravity of the platform. The second group corresponds to the heave plates and columns, and calibration forces are applied in the center of the heave plates assuming a dominant effect of these over the cylinders. Table 2 provides a summary of the coefficients of the damping terms obtained in the
calibration phase.

Table 2: Model calibration: added mass, linear damping and quadratic damping coefficients.

<table>
<thead>
<tr>
<th>Applied to CG</th>
<th>B_{11}</th>
<th>75  KN(m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surge linear damping</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Heave added mass</td>
<td>A_{33}</td>
<td>1200 Tn</td>
</tr>
<tr>
<td>Heave linear damping</td>
<td>B_{33}</td>
<td>110 KN(m/s)</td>
</tr>
<tr>
<td>Heave quadratic damping</td>
<td>B_{23}^2</td>
<td>805 KN(m/s)²</td>
</tr>
</tbody>
</table>

Applied at the center of heave plates

| Heave linear damping                   | B_{33} | 75  KN(m/s) |
| Heave quadratic damping                | B_{23}^2 | 805 KN(m/s)² |

Figure 3 shows experimental versus numerical results after calibration for the surge, heave, and pitch decay tests. Good agreement has been reached for the three degrees of freedom. The natural periods obtained are 70 s in surge, 19 s in heave and 26 in pitch respectively.

Figure 3: Comparison between experimental and numerical decay tests.

3.3 Regular waves

One the model has been calibrated against decay test the Response Amplitude Operators (RAOs) of the computational model are obtained and compared with those obtained in the experiments. Three catenary lines were used as mooring lines for the rest of the experiments. Mooring line particulars are given in Table 1.

It has to be said that the experimental data shows a change in the platform response along the experiment, it is to say, the results of the spectral analysis depend on the time interval used. If the period of time used for calculating the RAOs is chosen towards the end of the experiment, an increase of the response in the low frequencies is observed which has raised the concern on whether longer waves are being dissipated appropriately by the beach located at the opposite end of the wave maker. Experimental RAOs were obtained for an equivalent wave height of $H = 2$ m. Figure 5 compares the RAOs in surge, heave, and pitch, respectively, against the numerical results. A good agreement has been found for the three degrees of freedom.
3.4 Bichromatic waves

A number of tests were carried out with bichromatic waves in order to analyze the second-order response of the platform. The incident wave periods range between 5.5 and 21 seconds, corresponding to wave lengths between 47 and 689 meters. The wave frequency difference ranges from 0.0145 Hz to 0.0152 Hz. The latter frequency is close, on purpose, to the surge resonant frequency since the focus of the analysis will be on the surge response to the slow drift.

Table 3 provides the experimental test matrix, including incident wave height (H), length (λ) and period of the two incident waves (T). All these cases have been simulated in the time-domain solver. Once the simulations were carried out, the time series have been transformed to the frequency domain in order to make easier the comparison of the results with the experimental data. Sampling frequency has been adjusted so that the incident wave frequencies, as well as the difference and the sum, are precisely captured by the results of the Fast Fourier Transform (FFT) analysis.

Table 3: Bichromatic wave test.

<table>
<thead>
<tr>
<th>Case</th>
<th>Incident wave 1</th>
<th>Incident wave 2</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>T1</td>
<td>H1</td>
</tr>
<tr>
<td>1</td>
<td>3.44</td>
<td>689</td>
</tr>
<tr>
<td>2</td>
<td>2.82</td>
<td>564</td>
</tr>
<tr>
<td>3</td>
<td>5.64</td>
<td>564</td>
</tr>
<tr>
<td>4</td>
<td>5.63</td>
<td>450</td>
</tr>
<tr>
<td>5</td>
<td>5.27</td>
<td>351</td>
</tr>
<tr>
<td>6</td>
<td>2.83</td>
<td>189</td>
</tr>
<tr>
<td>7</td>
<td>1.88</td>
<td>94</td>
</tr>
<tr>
<td>8</td>
<td>1.67</td>
<td>84</td>
</tr>
<tr>
<td>9</td>
<td>1.35</td>
<td>67</td>
</tr>
<tr>
<td>10</td>
<td>1.22</td>
<td>61</td>
</tr>
<tr>
<td>11</td>
<td>1.11</td>
<td>55</td>
</tr>
</tbody>
</table>

A comparison of the platform movements between the experimental data with the numerical results obtained in this work is given in Figures 5, 6, 7. These figures show the movement...
amplitude obtained by applying FFT analysis to time domain response. Very good agreement is found in the incident wave frequencies, which basically correspond to first-order response. Regarding to second-order response, a good agreement is also found between the numerical and experimental results, as well as for the rest of frequencies. In fact, the numerical results follow quite well the experimental trends. It has to be mentioned that measuring second-order response in experiments is a tough task, and can easily introduce uncertainties in the final results.

Figure 5: Case 1, 2, 3 and 4. Pitch (on the right), heave and surge (on the left) response amplitudes.
Figure 6: Case 5, 6, 7 and 8. Pitch (on the right), heave and surge (on the left) response amplitudes.

Figure 8 compares the experimental and numerical tensions registered at the fairlead point for the line 1. This mooring line represents the mooring line aligned with the direction of wave propagation. The lines 2 and 3 are placed in a symmetric configuration with respect to the wave direction. Since numerical simulation were carried out without sway, roll, and yaw due to the symmetry of the problem), numerical results for mooring lines 2 and 3 are the same. The results are not relevant compared with line 1. The experimental measurements of the fairlead mooring tension and the values obtained from the numerical simulation tensions compare well considering...
all the uncertainties of the experiment, as well as the difficulty of measuring second-order effects. Looking at the numerical results in Figure 8, it is observed that mooring model provide very similar results.

4 CONCLUSIONS

A FEM model for the second-order wave diffraction-radiation problem in the time-domain has been validated against experiments carried out at the ECN Nantes facilities for the HiPRWind model. In a first stage, the model viscous damping was calibrated to reproduce decay tests for surge, heave, and pitch. A good match between the experiments and the calibrated model was obtained. Then, RAOs were compared using the monochromatic wave tests, finding that the numerical results follow well the trend of the experiments. Finally, simulation of the platform subject to bichromatic waves were performed and compared to the experiments. Taking into account the experimental uncertainties associated to measuring second-order quantities, it has been found that the computed movements of the platform reproduced well the experimental data.
Regarding the modeling of the mooring lines, the obtained results agree well with the experimental trends. They also indicate that for the operational conditions analyzed in this work the nonlinear FEM dynamic cable provides quite similar results. In conclusion, the second-order time-domain FEM model along with the nonlinear mooring model has been proven to successfully solve the second-order movements of the HiPRWind platform. In particular, under bichromatic waves designed to enhance slow-drift problem in surge.

5 ACKNOWLEDGEMENTS

The authors acknowledge ECN Nantes which facilities made this work possible. The authors thank A.N. Simos, F. Ruggeri, R. Watai, A. Souto-Iglesias and C. Lopez-Pavon for providing the experimental results for the semi-submersible floater. Thanks also to Acciona Energa and Fraunhofer Institute, and especially to Raul Manzanas. The research leading to these results
has received funding from the Spanish Ministry for Economy and Competitiveness under Grants ENE2014-59194-C2-1-R and ENE2014-59194-C2-2-R (X-SHEAKS). The authors also gratefully acknowledge the support of NVIDIA Corporation with the donation of a TeslaK20 GPU used for this research.

REFERENCES


NUMERICAL SIMULATION OF AN ARRAY OF HEAVING FLOATING POINT ABSORBER WAVE ENERGY CONVERTERS USING OPENFOAM

BRECHT DEVOLDER1,2, PIETER RAUWOENS2 AND PETER TROCH1

1 Ghent University  
Department of Civil Engineering  
Technologiepark 904, 9052 Ghent, Belgium  
e-mail: Brecht.Devolder@UGent.be, Peter.Troch@UGent.be

2 KU Leuven  
Technology Campus Ostend, Department of Civil Engineering  
Zeedijk 101, 8400 Ostend, Belgium  
e-mail: pieter.rauwoens@kuleuven.be

Key words: Wave energy, floating point absorber, array, CFD modelling, experimental validation

Abstract. In this paper we use the CFD toolbox OpenFOAM to perform numerical simulations of multiple floating point absorber Wave Energy Converters (WECs) in a numerical wave basin. The two-phase Navier-Stokes fluid solver is coupled with a motion solver to simulate the wave-induced rigid body heave motion. The key of this paper is to extend numerical simulations of a single WEC unit to multiple WECs and to tackle the issues of modelling individual floating objects close to each other in an array lay-out. The developed numerical model is validated with laboratory experiments for free decay tests and for a regular wave train using two or five WECs in the array. For all the simulations presented, a good agreement is found between the numerical and experimental results for the WECs’ heave motions, the surge forces on the WECs and the perturbed wave field. As a result, our coupled CFD–motion solver proofs to be a suitable and accurate toolbox for the study of wave-structure interaction problems of multiple floating bodies in an array configuration.

1 INTRODUCTION

Wave energy from ocean waves is captured by Wave Energy Converters (WECs) and converted into electrical power. In this study, WECs of the floating point absorber (FPA) type are selected. The numerically obtained viscous flow field around and the response of a single WEC unit have been validated with experimental data in previous work of the authors [1]. Now, this study focusses on the hydrodynamics around and the response of a small array of two and five WECs respectively. However, this is a starting point for wave farm modelling in which the interaction between a large number of closely spaced WECs will be analysed.

The Computational Fluid Dynamics (CFD) toolbox OpenFOAM [2] is used to study array effects in a numerical wave basin by solving the three dimensional flow field around and the
The main focus of the paper is put on the numerical simulation of several free decay tests using different array layouts. One WEC unit is initially placed out of equilibrium and released, leading to a damped oscillatory motion until all the forces acting on that WEC are in equilibrium. Moreover, the motion of the WEC generates radiated waves with decreasing wave heights away from the WEC. Those radiated waves initiate the motion of and a surge force on the neighbouring WECs in the array. Simulations are performed in order to compare the radiated wave field, the motion of and the surge force on the heaving WECs with experimental data measured in a wave basin. The purpose of the simulations is to demonstrate the ability of the coupled CFD–motion solver to simulate wave propagation of the radiated wave field in an array.

The second part of the paper is dedicated to present CFD simulations of an array of two WEC units subjected to a specific regular wave train. Only the heave motion of the WECs is considered and together with the perturbed wave field validated against laboratory results.

The capability of OpenFOAM to study wave-body interactions is already reported by [3]. An excellent description and comparison of the different numerical models for wave energy devices is provided in [4]. They mentioned that good agreements have been obtained between CFD and experimental results, demonstrating the feasibility of CFD simulations for wave energy applications. As mentioned before, simulations of a single WEC unit have been reported in previous work of the authors [1] but also in [5,6]. Numerical simulations of WEC arrays using simplified radiation-diffraction models have been published in [7–9]. However, CFD simulations of a WEC array have only been reported by a few researchers, e.g. [10,11].

2 EXPERIMENTAL SETUP

In this study, experimental data is used from the WECwakes project [12] conducted in the shallow water wave basin of the Danish Hydraulics Institute (DHI; Hørsholm, Denmark). Up to 25 WEC units were installed in the basin, which has a length of 25 m and a width of 35 m. For all the tests, a constant water depth of 0.70 m was maintained in the basin.

The WEC’s geometry is depicted in Figure 1. The WECs are characterised by a mass \( m \) of 20.545 kg, a total height \( h_{WEC} \) of 0.60 m, a diameter \( D \) of 0.315 m and a draft \( d_{WEC} \) of 0.315 m. At the top of the WEC, the power-take off (PTO) system is installed (Figure 1). The PTO force is applied to the buoy by mimicking a Coulomb damper using friction brakes (composed of two PTFE-blocks and four springs) between the float and the supporting axis.

The enormous experimental database is of large interest for the validation and extension of different numerical models. In this paper, our coupled CFD–motion solver is validated by using the available experimental dataset generated during the WECwakes project.

3 NUMERICAL FRAMEWORK

Numerical modelling is performed for the study of individual WEC units configured in an array layout. The two-phase flow solver with dynamic mesh handling, interDyMFoam, is available in OpenFOAM to perform simulations of moving bodies installed in a numerical wave basin.
3.1 Flow solver

Simulations of the two-phase flow field are performed by solving the incompressible RANS-equations, with a conservation equation for the Volume of Fluid (VoF) [13]. Turbulent effects are not dominating since the flow of the simulations presented is always characterised by a low Keulegan-Carpenter (KC) number. Therefore in the first instance, only laminar solutions are generated. However, in case turbulence plays a role, we refer to [14] on how to properly deal with turbulence near the air-water interface.

For all simulations the following settings are used: central discretisation for the pressure gradient and the diffusion terms; TVD (total variation diminishing) schemes with a van Leer limiter [15] for the divergence operators; second order, bounded, implicit time discretisation; a maximum Courant number of 0.30.

3.2 Computational domain

All the numerical simulations are performed in a numerical wave basin which represents the experimental wave basin as good as possible. However, some simplifications are made in order to obtain reasonable simulation times. Firstly, a vertical symmetry plane through the centre of the WECs is implemented over the length over the basin, as indicated in Figure 2 for the 5WEC-array. This is justified because the WECs tested in this paper are all installed in the middle of the basin and no asymmetric effects are expected (low KC numbers). For the free decay test using five WECs, a symmetrical heave motion of WEC2 and WEC4 and WEC1 and WEC5 is expected because the middle WEC (WEC3) is decaying and the spacing between the individual WECs is fixed to 5D. Therefore a second symmetry plane is implemented perpendicular to the first symmetry plane and through the centre of WEC3 (see Figure 2). Each boundary of the computational domain needs specific boundary conditions.
We use the IHFOAM toolbox [16,17] to implement the wavemaker and absorbing beach in the experimental facility. The side wall of the numerical wave basin is sufficiently far enough from the array, > 8D, to neglect its influence on the hydrodynamics around the WECs.

Figure 2: Plan view of the numerical wave basin for the 5WEC-array using two symmetry planes during a free decay test of WEC3. The red dots indicate the position of three wave gauges: WG9, WG10 and WG11.

The numerical wave basin is represented by a structured grid consisting of only hexahedral cells with local refinements in the zones of interest (i.e. around the free water surface and the WECs). A longitudinal cross section of the numerical domain around the WEC array during a free decay test is depicted in Figure 3 for the 5WEC-array. The vertical grid resolution is about 1 cm in the zones of interest, which is sufficiently according to [1]. The horizontal cell size increases towards the boundaries of the wave basin in order to limit the number of cells. The only exception is for the simulation where regular waves are generated at the inlet, in that case the horizontal cell size is kept constant towards the inlet boundary only. The high aspect ratio observed for the cells above the decaying WEC is explained in the next paragraph.

Figure 3: Cross section of the computational domain for the 5WEC-array (WEC3, WEC4 and WEC5), using two symmetry planes, during the initial condition of the free decay test of WEC3 (blue = water, grey = air).

3.3 Rigid body motion

The CFD-fluid solver is coupled with a motion solver in order to simulate rigid body motions. Only the governing motion of the WEC’s behaviour is considered, the heave motion. During each time step in the transient simulation, an iterative procedure is needed to obtain a converged solution for both the fluid solver and the motion solver. We developed a method that accelerates this coupling procedure and hence reduces the amount of sub-iterations for each time step to four. The key ingredient of this method is a good estimator for the WEC’s hydrodynamic added mass [18].

A second order accurate Crank-Nicolson integration scheme is used to derive the new position of the WEC from its acceleration $a$. The acceleration itself is based on Newton’s
second law: \( F = m \cdot a \) in which the force \( F \) is the sum of the pressure, shear and gravity forces acting on all the boundary faces of the WEC calculated with the fluid solver. The WEC’s mass \( m \) is determined using the procedure developed in [1].

In order to simulate multiple independently moving WECs in an array configuration, arbitrary mesh interfaces (AMIs) are implemented in order to create sliding meshes (see dashed vertical lines in Figure 4 for the case of two WEC units). These AMIs define a zone of cells around each WEC unit. In each zone, only the lowest and highest row of cells (see blue shaded boxes in Figure 4) are expanded or compressed according to the motion of the WEC unit located in that zone. This is implemented to prevent undesirable mesh deformation around the air-water interface, enhancing the accuracy of the solution. As a consequence, high aspect ratios are obtained for the distorted cells at specific time instants. However, those cells are not inside the zones of interest and will therefore not affect the accuracy of the simulations. All the variables solved with the flow solver, such as velocity, pressure and volume fraction, are interpolated over the AMIs.

**Figure 4:** A definition sketch of two independently moving WECs inside a three-dimensional computational domain of hexahedral cells. Only the highest and lowest row of cells (blue shaded boxes) in a zone are distorted (expanded or compressed) according to the heave motion of the WEC located in that zone. In between the zones, AMIs are implemented to create sliding meshes (dashed lines).

## 4 RESULTS AND DISCUSSION

Numerical simulations of a small array are performed for two free decay tests and a regular wave train. Firstly, a free decay test is performed for both a 2WEC-array and a 5WEC-array by pushing one WEC down, release it instantaneously and monitor the response of the WEC itself, and the neighbouring WEC(s). Secondly, regular waves are generated to obtain the response of a 2WEC-array and the resulting perturbed wave field.
4.1 Free decay test using two WECs in a line, spacing = 5D

During this first test using a 2WEC-array, WEC5 is lifted higher than its equilibrium position, released, and a free decay test is started. At a distance of $5D = 1.575$ m, WEC4 is freely floating in the water and will heave due to the radiated waves generated by WEC5.

In order to tune the numerical decaying motion to the experimental data, the methodology as reported in [1] is applied, resulting in a linear damper with a damping coefficient of 1.86 kg/s for WEC5. For WEC4, a much larger damping coefficient equal to 40 kg/s is used (see next section using the 5WEC-array). The resulting heave motions for WEC5 (decaying) and WEC4 (freely floating) are presented in Figure 5. It is clearly shown that the decaying motion of WEC5 is identical for the numerical and experimental model during the first 10 seconds. Thereafter, some small discrepancies are observed due to damping nonlinearities present during the experiments. It is however difficult to measure experimentally small heave motions due to friction of the bearings along the steel shaft (cfr. the WECwakes experiments). For WEC4, the resulting numerical heave motion is very small compared to the heave motion of WEC5. Unfortunately, the experimental recording failed for this test.

![Figure 5](image)

**Figure 5:** Vertical position of WEC5 (top) and WEC4 (bottom) during a free decay test of WEC5 with respect to its equilibrium position ($z_{WEC} = 0$ m) obtained with CFD (blue line) compared to experimental data (red line).

Additionally, the surge (horizontal) force on WEC4 due to the radiated wave field is compared between the numerical and experimental model and depicted in Figure 6. Again, a good comparison is found between both models. It is important to note that we filtered out the noise in the time signals of the experimental force measurements using a bandpass filter.

![Figure 6](image)

**Figure 6:** Surge force acting on WEC4 during a free decay test of WEC5 obtained with CFD (blue line) compared to the experimental determined surge force using two load cells after filtering the noise (red line).

Finally, the radiated wave field is given in Figure 7 for both numerical and experimental
data using the three wave gauges shown in Figure 2. The maximum observed amplitude of these radiated waves is smaller than 1 cm. Despite these small-amplitude waves, both results are very similar. In the first 10 seconds of the signals, the amplitude as well the phase of the radiated wave field is modelled close to the experimental results. Thereafter, some deviations between both results are observed due to the different behaviour of the numerical and experimental boundary conditions responsible for the absorption of the radiated waves.

![Figure 7: Radiated wave field around the 2WEC-array during a free decay test of WEC5 obtained with CFD (blue line) compared to the experimental measurements (red line).](image)

4.2 Free decay test using five WECs in a line, spacing = 5D

The next test comprises a free decay test of a 5WEC-array in which WEC3 is lifted higher than its equilibrium position, released, and a free decay test is started. As presented before in Figure 2, the 5WEC-array is simplified to a 2.5WEC-array using two symmetry planes. WEC4 and WEC5, at a distance of 5D and 10D respectively, are freely floating in the water and will move due to the radiated waves generated by the decaying motion of WEC3.

The vertical position of WEC3 (decaying) and WEC4 and WEC5 (freely floating) are depicted in Figure 8 for both numerical and experimental data. As motivated in the previous paragraph, a linear damper is used for the three WECs simulated using a damping coefficient of 1.86 kg/s, 40 kg/s and 100 kg/s for WEC3, WEC4 and WEC5 respectively. The same conclusions are drawn for the decaying WEC3 as reported in the previous paragraph: a very good agreement is found between the numerical and experimental signal. Based on Figure 8, the values used for the linear damper of WEC4 and WEC5 are sufficiently accurate to obtain the same amplitudes in the time signal for both the numerical and experimental data. Interestingly, the damping coefficients for WEC4 and WEC5 are significantly larger than the one used for the decaying WEC3. As reported in [11], this is needed to take the influence of nonlinear stiction damping effects and bearing friction along the steel shaft into account for
small heave motions. It is also shown that the motion of WEC4 is closer to the experimental data compared to the motion of WEC5. This is due to the larger distance of WEC5 from the decaying WEC. Moreover, the experimental data reveal that the behaviour of WEC2 and WEC4 and WEC1 and WEC5 is not fully symmetrical. Additionally, the surge force on WEC4 and WEC5 due to the radiated wave field is compared between the numerical and experimental model and visualised in Figure 9. Again, a very good comparison is found between both models. Subsequently, the radiated wave field is given in Figure 10 for both numerical and experimental data using three wave gauges (see Figure 2). The maximum observed amplitude of these radiated waves is again smaller than 1 cm and a fair agreement is found between the numerical and experimental time series.

Figure 8: Vertical position of WEC3 (top), WEC4 (middle) and WEC5 (bottom) during a free decay test of WEC3 with respect to its equilibrium position ($z_{\text{WEC}} = 0$ m) obtained with CFD (blue line) compared to experimental data (green and red lines).

Figure 9: Surge force acting on WEC4 and WEC5 during a free decay test of WEC3 obtained with CFD (blue line) compared to the experimental determined surge forces after filtering the noise (green and red lines).
4.3 Regular waves test using two WECs in a line, spacing = 5D

The last test presented in this paper includes the generation of regular waves and subject them to the 2WEC-array. The waves have a height $H$ equal to 0.074 m, a wave period $T$ of 1.26 s and are generated in a water depth $d$ of 0.70 m. At the inlet, waves are generated using a second order Stokes theory and active wave absorption is turned on. For this simulation, a linear damper is used for both WECs with a damping coefficient of 1.86 kg/s. Moreover, a coulomb damper on each WEC is included because the PTO system was on during the experimental test. The PTO force is implemented in the numerical model as described in [12]:

$$ F_{PTO} = -\mu F_{spring} \text{sign}(v(t)) = -\mu 4 dx k_{spring} \text{sign}(v(t)) $$

(2)

where $v(t)$ is the WEC’s vertical velocity, $\mu = 0.17$, $dx = 30.5$ mm and $k_{spring} = 0.14$ N/mm.

The heave motions of both WECs are visualised in Figure 11 for the numerical and experimental model respectively. It is observed that the numerical obtained heave motions are significantly larger, about 60 %, than the experimental results. Moreover, there is a time shift present in the signals for both WECs. Figure 12 presents the surge force acting on both WECs when subjected to a regular wave train. In contrast as observed for the heave motions, the numerical obtained surge forces are very similar to the experimental data. Lastly, the perturbed wave field (i.e. incident + diffracted + radiated wave field) is given in Figure 13 for both numerical and experimental data using three wave gauges (see Figure 2). The time signals confirm that an identical wave field is present in the numerical wave basin as observed during the experimental tests.
Figure 11: Vertical position of WEC4 (top) and WEC5 (bottom) during a regular wave test ($H = 0.074$ m, $T = 1.26$ s, $d = 0.70$ m) obtained with CFD (blue line) compared to the experimental heave motions (red line).

Figure 12: Surge force acting on WEC4 (top) and WEC5 (bottom) during a regular wave test ($H = 0.074$ m, $T = 1.26$ s, $d = 0.70$ m) obtained with CFD (blue line) compared to the experimental measurements after filtering the noise (red line).

As a conclusion, only a different behaviour in the WECs’ heave motions (amplitude + time shift) is observed between numerical and experimental data. Therefore, we assume that those discrepancies are mainly related to the different behaviour of the PTO system between the numerical and experimental model, which needs further investigation.

5 RESEARCH TOPICS UNDER INVESTIGATION

The topics listed below will be investigated in the near feature:
- The sensitivity of the linear damper acting on each WEC unit;
- The PTO force needs to be implemented precisely in the numerical model for an accurate representation of the spring system’s behaviour used during the experiments;
- Regular waves cause a net horizontal force acting on the WEC units inducing an additional vertical damping force (coulomb damper) apart from the PTO force applied;
- Including turbulent effects;
- Simulations of different numbers of WECs arranged in various layouts.
Figure 13: Perturbed wave field around the 2WEC-array during a regular wave test ($H = 0.074$ m, $T = 1.26$ s, $d = 0.70$ m) obtained with CFD (blue line) compared to the experimental data (red line).

6 CONCLUSIONS

We have presented several cases of numerical simulations of two and five heaving WECs installed in an array lay-out in a numerical wave basin. Regarding the free decay tests, a very good agreement is obtained between numerical and experimental results for both a 2WEC-array and a 5WEC-array. Not only the vertical position of the WECs and the surface elevations of the radiated wave field have shown an excellent agreement but also the surge force acting on the neighbouring WECs. Furthermore, simulations of an array of two WECs subjected to a specific regular wave train have returned promising results for its heave motion, the surge force on the WECs and the perturbed wave field around the WECs.

The numerical results have shown that our coupled CFD–motion solver is a robust and suitable toolbox to study wave-structure interaction. Moreover, the coupled model is accurate to analyse the interaction between multiple WECs installed in an array configuration. In particular, the surge force on the WECs and the perturbed wave field have been modelled very well. Future improvements will include a more accurate modelling of the PTO system to enhance the prediction of the WECs’ heave motion.

7 ACKNOWLEDGEMENTS

The first author is Ph.D. fellow of the Research Foundation – Flanders (FWO), Belgium (Ph.D. fellowship 1133817N).

The WECwakes project is funded by the EU FP7 HYDRALAB IV programme (contract no. 261520). The project is a consortium of seven European partners coordinated by Ghent University-Belgium (Peter Troch; Vasiliki Stratigaki). The construction of the WEC models at Ghent University (research grant FWO-KAN-15 23 712 N) and part of the data pre- and
post-processing (Ph.D. funding grant of Vasiliki Stratigaki and research project no. FWO-3G029114), have been funded by the Research Foundation Flanders, Belgium (FWO).

REFERENCES


THE EFFECTS OF TURBULENCE INTENSITY ON THE DOWNSTREAM PERFORMANCE OF HORIZONTAL AXIS TIDAL STREAM TURBINES.

I. Masters*, A. J. Williams*, M. Edmunds*, P. Pyakurel† J. H. VanZwieten†

*Marine Energy Research Group, Energy and Environment Research Group, Zienkiewicz Centre for Computational Engineering, College of Engineering, Swansea University, Swansea, Wales, UK.
E-mail: I.Masters@swansea.ac.uk, web page: http://www.swansea.ac.uk

† Florida Atlantic University
Florida, USA.
E-mail: jvanzwi@fau.edu, web page: http://www.fau.edu

Key words: Computational methods, marine engineering, tidal stream turbine, marine currents, turbulence, wake, computational fluid dynamics

Abstract. This study focuses on a comparison of model results from a blade element momentum computational fluid dynamics (BEM-CFD) steady state RANS model, and a BEMT model that accounts for the wake generated from upstream turbines using analytic expressions. Rotor forces are calculated using 3D hydrofoil profile data. Both techniques are validated against existing experimental data, and then used to assess the power extraction of downstream turbines. Turbulent inflow conditions of 3% and 7% are applied, and the results of power extraction assessed. Particular attention is paid to the velocity field, with respect to the downstream wake, to assess how the turbulence characteristics effect the recovery rate and downstream power potential. Both models highlight the different recovery rates of the two turbulent conditions.

1 INTRODUCTION

This study focuses on two approaches for the prediction of tidal stream energy extraction. One approach is the BEM-CFD steady state RANS model [1] which computes the rotor forces acting on the fluid from lift, drag, and geometric data supplied to the model as inputs. The velocity and turbulence fields are also predicted. The second approach utilises a BEMT model [2] that accounts for the wake generated from upstream turbines using analytic expressions for mean flow and turbulent wake properties. Rotor forces are calculated using 3D lift and drag, and a dynamic wake model is implemented.

To study the effects of turbulence on performance, the models are validated against existing experimental data [3], and then used to assess the power extraction of downstream turbines.
range of turbulent inflow conditions are applied, and the results of power extraction assessed. Particular attention is paid to the velocity field, with respect to the downstream wake, to assess how the turbulence characteristics effect the recovery rate and downstream power potential.

The remainder of this paper describes the models, case setups, results, and conclusions. Section 2 describes the BEMT analytical wake field expression, this is then followed in Sections 3.1 and 3.2 with a discussion of the BEM-CFD model. In Section 4 the modelled cases are described. The results are then discussed in Section 5, and are followed by conclusions in Section 6.

2 Analytical Wake Field Expression

Empirical expressions have been developed to approximate the wake field behind horizontal axis tidal turbines in the far-wake region. These expressions represent the reduced flow speed using expressions originally developed for wind turbines [4], with coefficients optimised using experimentally measured flow speeds behind horizontal axis tidal turbine [5]. These expressions define velocity deficit in the far-wake as a function of thrust coefficient and ambient turbulence intensity [5]. Expressions for downstream turbulence intensities have also been developed and tuned by [5]. These expressions were not directly derived from wind wake models, but were created such that they are a function of the same two operating parameters; thrust coefficient and ambient turbulence intensity. For lower ambient turbulence intensities, such as the 3% and 7% ambient values used in this study, it was shown by [5] that expressions based on the Larsen model [6] are well suited for calculating centreline velocity deficit and those based on the Ainslie model [7] effectively describe the dependence on radial location. This model is referred to as the Larsen/Ainslie model by [4].

The Larsen/Ainslie expression described by [4] defines velocity deficit in the far wake region according to:

\[
\frac{U_w}{U_o} = 1 - U_c^* e\left(-3.56\left(\frac{r}{2r_o b}\right)^2\right)
\]

where \( r \) is the radial distance from turbine centreline, \( r_o \) is the radius of rotor, \( U_o \) is the free stream velocity,

\[
b = \left[\frac{3.56C_T}{8U_c^*(1 - 0.5U_c^*)}\right]^{\frac{1}{2}}
\]

and

\[
U_c^* = \frac{1}{9}\left(C_T A_d x^{-2}\right)^{\frac{1}{2}}\left(\frac{35}{2\pi}\right)^{\frac{3}{10}}\left(3c_1^2\right)^{-\frac{1}{5}}
\]

In these equations \( C_T \) is the thrust coefficient, \( A_d \) is the rotor area, \( x \) is the distance in the downstream direction (x axis), and \( c_1 \) is non-dimensional mixing length (empirical coefficient) defined by [5] as:

\[
c_1 = 0.0406 e^{0.1361 TI}
\]

where \( TI \) is the ambient turbulence intensity. It is suggested by [5] that this algorithm only be utilised within the wake radius suggested by Jensen [8] as:

\[
r_x = r_o + \alpha x
\]
Figure 1: Blade element method rotor discretisation scheme.

where $\alpha = 0.00003 TI^4 - 0.0009TI^3 + 0.0097TI^2 - 0.0396TI + 0.0763$. After this wake radius it is suggested that ambient flow speeds should be utilised.

The empirical expression for calculating the turbulence intensity in the far-wake region suggested by Pyakurel [5] defines turbulence intensity according to:

$$TI_w = TI \left( \frac{TI_c - TI}{TI} \right) e^{-3 \left( \frac{r}{D} \right)^2 + 1}$$  \hspace{1cm} (6)

where $TI_c$ is the centreline turbulence intensity calculated according to:

$$TI_c = \sqrt{\Delta TI^2 + TI^2}$$  \hspace{1cm} (7)

In this equation the increased turbulence intensity at the centreline is calculated from:

$$\Delta TI^2 = \frac{1.5}{TI^{0.15}} C_t^{0.4} \left( \frac{x}{D} \right)^k$$  \hspace{1cm} (8)

where $k = -2TI^{0.1}$.  

3 BEM-CFD

This section introduces the hybrid analytical, BEM, and CFD computational model. First is a short discussion of the CFD process, governing equations and turbulence models (Section 3.1). The following section outlines the BEM-CFD approach (Section 3.2), as described by [1], and validated against [3].

3.1 CFD and the Governing Equations

CFD simulations are conducted using OpenFOAM [9]. Linking the CFD flow domain to the BEM model is achieved by additional source terms included within the conservation of momentum equations of the solver. The solver uses steady state Reynolds averaged incompressible Navier-Stokes equations with a range of turbulence model options including K-Epsilon, K-Epsilon RNG, and K-Omega.
The CFD model requires the solution of the Navier Stokes equations representing the conservation of mass and momentum. These equations are expressed as follows:

\[ \nabla \cdot (\rho \vec{v}) = 0 \]  

\[ \nabla \cdot (\rho \vec{v} v_i) = -\frac{\partial p}{\partial x_i} + \nabla \cdot ([\mu_l + \mu_t] \nabla [\rho v_i]) + S_i \]  

where \( \rho \) is the density, \( \vec{v} \) is the velocity vector, \( v_i \) is the \( i \)th component of the velocity vector, \( \mu_l \) and \( \mu_t \) are the laminar and turbulent dynamic viscosities respectively, and \( S_i \) includes an additional source representing the moving rotor.

A widely used method for simulating the effect of turbulence on the mean flow, at the sub grid level, is the \( k-\epsilon \) turbulence model [10]. Although this model is relatively simple, stable, and requires modest computational cost, it is limited by the single length scale used to calculate the viscous properties of the turbulent fluctuations. Diffusion, as a result of the turbulent fluctuations, does not occur at just one length scale. The \( k-\epsilon \) RNG model [11] tries to address this issue by utilising statistical analysis to describe the set of turbulent length scales. Although these models regard turbulence as being isotropic in nature, in rotational flow, such as found in turbine wakes, the turbulent eddies are likely anisotropic. In this work the focus on performance of turbine rotors, rather than flow structures of the highly turbulent near wake region immediately downstream of the device, justifies the use of the \( k-\epsilon \) RNG model.

In this model two equations are solved; \( k \) represents the energy contained within the turbulent fluctuations, and \( \epsilon \) represents the dissipation rate of this energy. The equations for the transport of these variables are similar in form to the momentum equations:

\[ \nabla \cdot (\rho \vec{v} k) = \nabla \cdot \left( \left[ \mu_l + \frac{\mu_t}{\sigma_k} \right] \nabla k \right) + \mu_t G - \rho \epsilon \]  

\[ \nabla \cdot (\rho \vec{v} \epsilon) = \nabla \cdot \left( \left[ \mu_l + \frac{\mu_t}{\sigma_\epsilon} \right] \nabla \epsilon \right) + C_{1\epsilon} \mu_t G \frac{\epsilon}{k} - C_{2\epsilon} \left( \frac{C_{\mu} \eta^3 [1 - \eta/\eta_0]}{1 + \beta \eta^3} \right) \]  

where:

\[ \eta = \sqrt{\frac{k}{\epsilon}} \]
I. Masters, M. Edmunds, A. J. Williams, P. Pyakurel and J. H. VanZwieten

Figure 3: BEMT analytical mean wake velocity in the far wake region behind horizontal axis tidal turbines for turbulence intensities of 3% (top) and 7% (bottom).

These equations are used to calculate a turbulent viscosity:

\[ \mu_t = \frac{\rho C_\mu k^2}{\epsilon} \]  

(14)

In Equations 11, 12, and 14; \( \sigma_k, \sigma_\epsilon, C_{1k}, C_{2k}, C_\mu, \beta, \) and \( \eta_0 \) are taken to be constants, and \( G \) represents the turbulent generation rate. The viscosity components of the \( k \) and \( \epsilon \) equation diffusion terms, are effectively the sum of the laminar and turbulent viscosities.

3.2 The BEM-CFD Method

Fluid flowing over the surface of a body has a force exerted on it. Of interest, when describing hydrofoils, are the lift and drag components of this force, see Figure 1. Lift is defined as a force perpendicular to the free stream flow direction. Drag is defined as being parallel to, and opposing, the free stream flow direction. These hydrodynamic forces can be described by the following equations:

\[ F_L = 0.5 \rho \bar{v}^2 AC_L \]  

(15)

\[ F_D = 0.5 \rho \bar{v}^2 AC_D \]  

(16)

where \( F_L \) is lift force, and \( F_D \) is drag force, \( \rho \) is fluid density, \( \bar{v} \) is velocity, \( A \) is area, \( C_L \) is the coefficient of lift, and \( C_D \) is the coefficient of drag. The coefficients of lift and drag are dependant on the angle of attack, \( \alpha \), the geometric properties of the hydrofoil, and the Reynolds Number. Data for a wide range of profiles is available from a number of sources including [12]. The chord and twist geometric properties, in combination with the profile \( C_L \) and \( C_D \) characteristics, are a function of the overall rotor design/performance requirements.

The influence of a tidal turbine with multiple hydrofoils (or blades) is averaged over a rotational interval, i.e. the rotor applies the same force to all locations at the same radial distance.
I. Masters, M. Edmunds, A. J. Williams, P. Pyakurel and J. H. VanZwieten

Figure 4: BEMT analytical turbulence intensity in the far wake region behind an horizontal axis tidal turbine for turbulence intensities of 3% (top) and 7% (bottom).

from the rotor centre on a given axial plane. The magnitude of these forces are a function of the hydrofoil geometry, its hydrodynamic properties, the quantity of hydrofoils, and their speed relative to the flow.

The BEM-CFD model is formulated such that the relevant characteristics of the hydrofoil are introduced through additional source terms appended to the momentum equation, see Section 3.1. Figure 1 shows how a turbine with three hydrofoils is discretised for use with the blade element method. The hydrofoil properties are determined at radius $r_i$, and then averaged over $2\pi$ radians. This process is repeated for each hydrofoil element over the interval $[r_0, r_{\text{max}}]$. For a detailed examination and validation of this approach, including tip loss correction, the reader is referred to [1].

4 Model Setup

The model described in Section 3.2 is implemented in OpenFOAM [9]. In this section is described the model setup including; mesh configuration, boundary conditions, and initial conditions, with respect to the OpenFoam Toolbox.

4.1 Case Studies

Investigations by [13, 14] evaluate horizontal axis tidal turbine performance, while studies by [15, 16, 17, 18, 19, 20] validate analytical or numerical models with experimental results. The work by [3] and [21] is of particular interest for the validation of this work. In this research the experimental work of [3] and [21] for a single and dual tidal turbines are compared with BEMT and BEM-CFD. The case studies are set such that the geometry of the turbines, and the domain extents, match that of the experimental studies.

In support of this study two turbulence cases are examined. Turbulence intensity (TI) is set at the inlets and allowed to dissipate to meet the required quantity at the centre rotor. The target TI for the two cases is 3% and 7% at the rotor. This required a TI setting of 5% and
I. Masters, M. Edmunds, A. J. Williams, P. Pyakurel and J. H. VanZwieten

Figure 5: Top images are of velocity magnitude, while the bottom images are of turbulence intensity at 3%. The left images are of the single rotor case, and the right images are of the dual rotor case. All plots are taken in the horizontal plane at hub height, and the axis units are in diameters.

8% respectively to be set at the inlet. A matrix of full TSR sweeps [1...7] is run for each of the turbulence and single/dual rotor cases. The front rotor for all 8 diameter spaced dual rotor cases is held at TSR4. A single run of TSR3.67 front and rear rotor is also run for each turbulence case at 4 diameter spacing.

4.2 Mesh Configuration

Using the blockMesh utility, the mesh domain is configured as a hexahedron with bounding points [-9, -2, -1] to [9, 2, 1] which captures the domain extents of 18m x 4m x 2m in x, y, and z [3]. The domain is then subdivided in by [150, 40, 20]. Simple grading is used to refine the mesh to 0.2m at the left, right, and bottom boundaries, see Figure 2.

Refinement of the mesh around the turbines and wake region is achieved using the snappy-HexMesh utility. The wake region is defined as a cylinder 0.7m radius, extending from 0.7m in front of the rotor to the end of the domain. The refinement level in this region is specified as level 2. The rotor assemblies are set with a refinement level of 3, see Figure 2.

Three arrangements of mesh are generated for this study. One has a single turbine at [0,0,0], one has 2 turbines where the second is set at 4 diameters (2.8m) downstream, and one has 2 turbines where the second is set at 8 diameters (5.6m) downstream. The boundary layer height at the turbine geometry is set to 0.2m, providing a cell centre height (0.1m) which achieves a $y+$ of $\approx 150$.

4.3 Initial and Boundary Conditions

The initial conditions (and inlet condition) for velocity is set to 0.8m/s. Fixed value of 0.0m/s are set at the boundaries except the top and outlet, which are set to slip and zero gradient respectively. Kinetic energy and dissipation rate are set similarly and use wall functions to compute the boundary layer. The kinetic energy initial condition and inlet is set to 0.0024$m^2.s^{-2}$ (for the 3% TI case) and 0.0061$m^2.s^{-2}$ (for the 7% TI case), while the dissipation rate initial condition and inlet is set to 0.000252$m^2.s^{-3}$ for both cases.
5 RESULTS

5.1 Analytical Wake Field Expression

The algorithms presented in Section 2 are utilised to calculate the mean flow field and turbulence levels behind horizontal axis tidal turbines operating in flows with ambient flow speeds of 0.8 m/s and turbulence intensity values of 3% and 7% (Figures 3 and 4). For these calculations a turbine thrust coefficient of 0.8 is assumed. This fits well with the BEM-CFD results, which show a thrust coefficient of 0.74 at a tip speed ratio (TSR) of 4 at the centre turbine. Since these equations model the far-wake field which typically starts around five diameters downstream only distances beyond this are shown. These figures show that for both turbulence intensities, the wake deficit persists much further downstream than the increased turbulence intensity values in the wake field. Figure 3 shows that the velocity in the far wake is lower and persists longer for lower turbulence intensities. It also shows that the wake field is narrower for lower turbulence intensities. Figure 4 suggests that at 5 diameters downstream the turbulence intensity values are actually greater for lower ambient turbulence intensities. It also suggests that turbulence intensity values nearly reach their ambient values at 13 diameters downstream for both of the evaluated ambient turbulence intensities. The power produced at eight diameter downstream is 39% of upstream power for experimental case [21], whereas the corresponding value for BEM-CFD model that allows TI dissipation is 37% and the analytic wake model is 46%.

5.2 BEM-CFD

The results for all cases converged with a residual less than 0.001. Convergence was achieved between approximately 300 and 1000 iterations depending on the case. All cases are decomposed into eight threads for processing, and total time to convergence ranged between approximately 300 to 900 seconds.

Figure 5 shows velocity magnitude and turbulence intensity plots at 0.8m/s inlet velocity, and 3% turbulence intensity measured at the centre rotor. The two cases shown in Figure 5 are single rotor running with a TSR of 3.67, and the dual rotor case where the centre rotor is set at a TSR of 4, and a rear rotor (8 diameters downstream) is set to a TSR of 3.67. It can
be observed that the velocity deficit behind the centre rotor is greater when the rear rotor is placed downstream. This phenomena is also observed in the TI plots. The single rotor plots compare favourably with the equivalent velocity and turbulence plots from [3], including the asymmetry of the wake caused by the interaction of the tower assembly with the wake rotation. This observation is not made in [1] as the assembly structure is not modelled.

Figure 6 shows the same case as in Figure 5, with the turbulence increased to achieve 7% at the centre rotor. The same trends can be observed for TI set to 7% as described for TI set to 3%. The notable differences include; a shorter wake recovery distance, and a movement upstream of the turbulence pooling in the centre of the wake. This improved wake recovery has a knock on effect on the downstream performance of the rotor.

The rotor performances shown in Figure 7 demonstrate the single rotor performance (Cp) compared to TSR and dual rotor performance (Cp of the downstream rotor) for the two turbulence cases. The upstream rotor is held at a TSR of 4. It is observed there’s a significant increase in downstream turbine performance at higher turbulence levels. This corresponds well with the trends observed in [21] when moving from 3% to 15% TI. It is noted that in [3] there is a small drop in performance when comparing the 3% case with the 15% case at the centre rotor, however, this is not observed in our results for the centre turbine when comparing 3% to 7% TI.

6 CONCLUSIONS

The BEM-CFD TSR sweep results (Figure 7) show good correlation with the experiments in [3] and [21]. The velocity and turbulence profiles in Figure 5 show similarly good correlation, which gives us confidence in the higher turbulence (TI = 7%) results shown in Figure 6. The results from the analytical wake field expressions show similar correlation (Figures 3 and 4) with the experimental results, and the BEM-CFD plots in 5 and 6.

From the results it can be seen that as free stream turbulence increases, the distance at which
the wake recovers to free stream velocity significantly reduces. This result is expected, however of note is that both models capture this with a useful level of accuracy. The CFD model provides a more asymmetric distribution of the velocity and turbulence fields, this is due to simulating the tower structure within the CFD domain. Another effect of this is the ‘pooling’ of higher turbulence at the 6 diameter downstream location (see Figures 5 and 6). This accumulation is the result of the tower drag wake mixing with the rotating turbine wake.

There is no drop in performance (although expected) at the front turbine when turbulence increases. This is likely due to the lift and drag curves not reflecting a change in characteristics as turbulence increases. This could also explain a small error in the rear turbine performance profile (Figure 7).

The BEMT analytical wake expression model provides a symmetric wake profile; the tower structure is not modelled in this type of simulation. The wake recovery and profile are consistent with experiment and BEM-CFD, demonstrating the effectiveness of this approach, and the future potential of computationally efficient turbine array prediction.

7 Acknowledgements

The Authors wish to acknowledge the financial support of the Welsh Assembly Government, the Welsh European Funding Office, and the European Regional Development Fund Convergence Programme. The work was also supported by the EPSRC funded “Extension of UKCMER Core Research, Industry and International Engagement” project (EP/M014738/1). The Author(s) acknowledge(s) the financial support provided by the Welsh Government and Higher Education Funding Council for Wales through the Sêr Cymru National Research Network for Low Carbon, Energy and Environment. (C001822). This material is based upon work supported by the National Science Foundation under Grant No. ECCS-1307889.

REFERENCES


A NUMERICAL STUDY ON THE SHEDDING FREQUENCY OF SHEET CAVITATION

Themistoklis Melissaris*, Norbert Bulten† and Tom van Terwisga◊

* Delft University of Technology
Wärtsilä Netherlands BV
Drunen, The Netherlands
e-mail: themis.melissaris@wartsila.com

† Wärtsilä Netherlands BV
Drunen, The Netherlands
e-mail: norbert.bulten@wartsila.com

◊ Delft University of Technology
Maritime Research Institute Netherlands (MARIN)
Wageningen, The Netherlands
e-mail: t.v.terwisga@marin.nl

Key words: Cavitation, Turbulent Viscosity, Shedding frequency, CFD, Hydrofoil

Abstract. The last decades there is a strong interest in predicting cavitation dynamics as it is a prerequisite in order to predict cavitation erosion. Industrial applications require accurate results in an acceptable time span and as a result there is a focus on large scale dynamics. In this paper the RANS equations are used to investigate the shedding frequency of sheet cavities in two-dimensional simulations. First a verification study is made for the NACA 0015 in 6 degrees angle of incidence. A grid sensitivity study is conducted in wetted flow and in steady (non-shedding) cavitating condition (σ=1.6). Then an investigation is conducted in order to capture the shedding frequency. The results show that only when a correction for turbulent viscosity at the cavity-water interface is used it was possible to capture the shedding frequency as found in other numerical studies. Furthermore, a validation study is conducted on a NACA66-312 α=0.8 for two different angles of attack. The obtained results are compared and validated with the experimental data from Leroux et al. They indicate that the 2D shedding frequency predicted by the numerical simulations is in good agreement with the frequency obtained in the experiment.
INTRODUCTION

Cavitation can be defined as the creation of vapour pockets in a liquid under very low pressures. In maritime industry the occurrence of cavitation is in general unavoidable and therefore accepted. However, it is important to know the extent where cavitation is not harmful in operation, both in terms of noise and erosion. As a result, there is a strong interest in the prediction of cavitation dynamics in the vicinity of a propeller in view of the assessment of cavitation erosion risk. Within Wärtsilä cavitation prediction on propellers and thrusters is already integrated in the design process and the use of this type of numerical simulations is continuously growing [1]. Nevertheless, there is a tendency to improve the accuracy and the credibility of the computations and to move a step forward to cavitation erosion prediction. The ultimate goal is to be able to predict the risk of cavitation erosion on marine propellers in behind condition with sufficient reliability.

Sheet cavitation may generate strong shock waves, originating from the collapse of the shed vapor structures, which then often lead to erosion of surface material. Existing literature (Bark et al. [2]) suggests that the natural shedding frequency sometimes plays a role in cavitation erosion. Erosion is primarily the result of an accumulated energy transfer from macro scale cavities to collapsing cavities close to a solid surface. However, it is not the energy that determines the erosion intensity, but the potential power which is converted into acoustic power produced by collapsing clouds [3, 4]. The released potential power is related to the cavity volume and the surrounding pressure. Both are considered important, however the shedding frequency of the cavity, namely the frequency of the generated pressure waves, is numerically more sensitive and can easily be compared with experimental data. For rotating parts (i.e. marine propellers) the importance of investigating the shedding frequency becomes higher as the natural frequency and the frequency of the external flow will lead to an amplification of the pressure pulses.

The development of cavitation on propeller blades and cavitation erosion has mostly a three-dimensional character. Nevertheless, in this study the flow in span-wise direction is considered as a secondary flow and two-dimensional simulations are conducted, as a first step, focusing on the numerical uncertainty. Three-dimensional simulations have been planned as a further stage of this study.

MATHEMATICAL MODELS

2.1 Governing equations

The equations solved are the Reynolds Averaged Navier-Stokes (RANS) equations, where each instantaneous quantity can be split into time-averaged and fluctuating components. An incompressible segregated flow model is selected solving the integral conservation equations of mass and momentum in a sequential manner combined with the SIMPLE pressure-velocity coupling algorithm. From the basic principles of conservation of mass and momentum, the governing equations are written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$

(1)
\[
\frac{\partial}{\partial t} (\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{uu}) = -\nabla p + \rho f + \nabla \cdot \tau \tag{2}
\]

where \( \mathbf{u} \) is the velocity tensor, \( \rho \) is the fluid density, \( \nabla p \) the pressure gradient, \( \rho f \) the exterior force density per unit mass and \( \tau \) the viscous part of the stress tensor.

The multiphase model used is the Volume of Fluid (VOF) method. Within the VOF approach, the fluid is treated as a single continuum, assuming a no-slip condition between liquid and vapor phases, with varying properties in space according to its composition \( a_v \). The volume fraction of the components is determined from the condition:

\[
a_v + a_l = 1 \tag{3}
\]

while density and viscosity are defined as:

\[
\rho = a_l \rho_l + a_v \rho_v \quad \text{and} \quad \mu = a_l \mu_l + a_v \mu_v \tag{4}
\]

### 2.2 Turbulence modeling

In this study the SST k-\( \omega \) turbulence model developed by Menter [5] is used in order to fully resolve the boundary layer. This approach effectively blends a k-\( \varepsilon \) model in the far field with a k-\( \omega \) model near the wall. Reboud and Delannoy [6] showed the important role of the re-entrant jet on the cavity break-off cycle. However, the use of this turbulence model leads to very strong turbulent viscosity in the cavity wake hindering the re-entrant jet formation. It is stated by Reboud et al. [7] that this effect, which is not representative of the real behaviour, has been analysed to be related to the hypothesis of homogeneous flow and its no-slip condition between the two phases. That no-slip condition behaves as an artificial increase of dissipation.

That problem has been treated by an empirical reduction of turbulence dissipative terms in the two-phase regions, by modifying the turbulent viscosity [7]:

\[
\mu_t = f(\rho)C_{\omega} \frac{k}{\omega}; \quad f(\rho) = \rho_v + \frac{(\rho_m - \rho_v)^n}{(\rho_l - \rho_v)^{n-1}}; \quad n \gg 1 \tag{5}
\]

where \( \rho_v \) is the vapor density, \( \rho_l \) the liquid density and \( \rho_m \) the mixture density. For the constant \( n \) a recommended value \( n = 10 \) has been used. This modification improves the numerical simulations by taking into account the influence of the local compressibility effects of the vapor/liquid mixture on the turbulent structure.

### 2.3 Cavitation modeling

The conservation equation that describes the transport of vapour is similar to the mass conservation for liquid and is described by:

\[
\frac{\partial a_v}{\partial t} + \nabla \cdot (a_v \mathbf{u}) = S_{a_v} \tag{6}
\]

In equation (6), \( S_{a_v} \) represents the source of volume fraction of vapor. In order to close the system of equations formed by the RANS equations and the transport equation for the vapor, a cavitation model should be introduced for the source term of the volume fraction of vapor. The cavitation model used is the model proposed by Schnerr-Sauer (2000) based on a simplified...
Rayleigh-Plesset equation, which neglects the influence of bubble growth acceleration, as well as viscous and surface tension effects:

\[
\frac{dR}{dt} = \text{sign}(p_v - p) \sqrt{\frac{2|p_v - p|}{3\rho_t}}
\]

(7)

Where \(p_v\) is the saturation pressure, \(p\) is the local pressure around the bubble and \(\rho_t\) is the fluid density. According to this rate the source term in equation (6) is defined as:

\[
S_{av} = \frac{4\pi R^2 n_0}{1 + \left(\frac{4}{3}\pi R^3\right) n_0} \frac{dR}{dt}
\]

(8)

3 CASE DESCRIPTION

Two hydrofoils are going to be calculated using RANS equations in StarCCM+. The one case is the NACA 0015, on which a verification study is conducted and the second one is the NACA66-312 \(\alpha=0.8\) as an interest to validate the results with experimental data. A velocity inlet boundary is used for the upstream flow and a pressure outlet is defined on the outlet boundary. The pressure on the outlet for every condition is derived by the cavitation number:

\[
\sigma = \frac{P_{out} - P_v}{\rho U_{in}^2/2}
\]

(9)

3.1 NACA 0015 computational domain

For the verification study the NACA 0015 hydrofoil is used at 6 deg angle of attack with a chord length \(c = 200\) mm and a computational domain as used in the VIRTUE WP4 Workshop [8]. The length of the domain is extended 2 and 4 chord lengths upstream and downstream of the foil respectively and in the \(y\) direction the wall is 285 mm away from the center of the foil (1400x570).

### Table 1: Grid features for the NACA 0015

<table>
<thead>
<tr>
<th>Grid</th>
<th># Cells</th>
<th>(y^+)</th>
<th>Level</th>
</tr>
</thead>
<tbody>
<tr>
<td>G1</td>
<td>25,059</td>
<td>1.2261</td>
<td>Coarse</td>
</tr>
<tr>
<td>G2</td>
<td>49,931</td>
<td>0.8407</td>
<td>Medium</td>
</tr>
<tr>
<td>G3</td>
<td>77,081</td>
<td>0.6808</td>
<td>Fine</td>
</tr>
<tr>
<td>G4</td>
<td>100,569</td>
<td>0.5829</td>
<td>Very fine</td>
</tr>
</tbody>
</table>

The 2D computational grid is derived from a 3D mesh using trimmed hexahedral cells with local refinements and prism layers along the walls (Figure 1). The grid generation process is
driven by specifying a base mesh size, relative to which all the sizes are defined (cell size in various regions, prism layer near wall thickness etc.). Finer meshes of the same topology are then automatically created by just reducing the base size. The only parameter that was kept constant was the prism layer total thickness so as to include the boundary layer in any case. The geometrical similarity was controlled then by changing the number of the prism layers. A grid sensitivity study is conducted following the approach as described in [9] using four different grids as indicated in Table 1.

The nominal speed during the simulations is 6 m/s corresponding to a Reynolds number based on the foil chord length equal to $1.09 \times 10^6$. A pressure that corresponds to two different cavitation numbers $\sigma=1.6$ and $\sigma=1.0$ is applied on the outlet resulting on a steady and an unsteady condition respectively. A no-slip condition is applied on the foil and a slip condition on the top and bottom wall (see Table 2). The number of inner iterations per time step, the time step and the order of temporal discretization are defined after a sensitivity study in the unsteady cavitating condition ($\sigma=1.0$). In the end, 40 inner iterations and a time step corresponding to Courant number 0.75 are selected using a second order temporal discretization scheme. The Courant number is defined as the product of the inlet velocity and the time step over the cell size in the x-direction:

$$\text{Courant} = \frac{V_{in} \cdot \Delta t}{\Delta x} \quad (10)$$

| Table 2: Boundary conditions and flow characteristics for the two hydrofoils |
|---------------------------------|-----------------|-----------------|-----------------|
| **Boundary Conditions**         | **NACA 0015**   | **NACA 66 mod $\alpha=0.8$** |
| Velocity inlet                  | 6 m/s           | 5.33 m/s        |
| Angle of incidence              | 6 deg           | 6 deg           | 8 deg           |
| Pressure Outlet                  | $P_{out}=20.9$ kPa ($\sigma=1.0$) | $P_{out}=31.7$ kPa ($\sigma=1.6$) | $P_{out}=16.5$ kPa ($\sigma=1.0$) | $P_{out}=20.5$ kPa ($\sigma=1.28$) |
| Turbulent Viscosity             | 1%              | 1%              |
| Turbulent Viscosity Ratio       | 10              | 10              |
| Reynolds Number                  | $1.09 \times 10^6$ | $0.8 \times 10^6$ |
| Water Temperature                | 24 °C           | 20 °C           |

### 3.2 NACA 66 computational domain

A second case has been used in order to validate the results with experimental data. The selected geometry is a modified NACA 66 series with a chord length $c = 0.150$ m, which can be referenced as “N1.1-mod. NACA 66(mod.)-312, $\alpha = 0.8$” tested by Leroux *et al.* [10]. The relative maximum thickness was $\tau = 12\%$ at 45% from the leading edge and the relative maximum camber was 2% at 50% from the leading edge. The exact coordinates of the tested hydrofoil can be found in [11]. In the experiment the hydrofoil was tested in four different angles of incidence, however simulations only for two angles are performed (see Table 7), one for a low shedding frequency (6 deg) and one for a higher shedding frequency (8 deg).

The same approach as in the NACA 0015 case has been used in order to validate the numerical set-up. The same grid topology is applied with a small refinement around the
hydrofoil due to the thinner shape (especially at the leading edge) leading to a coarse grid with 28,593 cells (Figure 2). A nominal free stream velocity of 5.33 m/s is applied to the inlet corresponding to a Reynolds number based on the foil chord length equal to 0.8x10^6. A pressure that corresponds to cavitation number equal to σ=1.0 and σ=1.28 is applied on the outlet for angle of incidence 6 and 8 deg respectively. No-slip condition on the foil and slip condition on the walls are applied. The number of inner iterations per time step, the courant number and the order of temporal discretization are identical as in the NACA 0015 case. All the details for both foils can be found on Table 2.

Figure 2: Refinement visualization around the NACA66 foil (left) and the applied computational domain (right)

4 RESULTS

4.1 NACA 0015

The flow around the frequently used NACA 0015 hydrofoil is investigated in three conditions: wetted flow, steady cavitating flow (σ=1.6) and unsteady cavitating flow (σ=1.0). The results in wetted flow are compared with experimental data [8] and the ones in cavitating flow only with other numerical works [13-18].

4.1.1 Wetted flow

A grid sensitivity study is conducted in wetted flow for the drag (Figure 3) and lift (Figure 4) coefficients and the results are compared with experimental data available from the VIRTUE Workshop [8]. The results are shown in detail in Table 3. An error estimation has been made by using an approach with power series expansion proposed by Eca and Hoekstra [9]. The expansions are fitted to the data in the least-squares sense.

Figure 3: Drag coefficient for different grid density and comparison with the experimental value.

Figure 4: Lift coefficient for different grid density and comparison with the experimental value.
Table 3: Drag and Lift coefficient values for all the grids and comparison with experimental values (Δexp). The uncertainty (Uφ) of the values is also shown.

<table>
<thead>
<tr>
<th>Grid</th>
<th>CD</th>
<th>Uφ</th>
<th>% Δexp</th>
<th>CL</th>
<th>Uφ</th>
<th>% Δexp</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>0.014</td>
<td>0.658</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>G1 (Coarse)</td>
<td>0.01459</td>
<td>8.47%</td>
<td>4.20%</td>
<td>0.67018</td>
<td>2.40%</td>
<td>1.85%</td>
</tr>
<tr>
<td>G2 (Medium)</td>
<td>0.01435</td>
<td>3.48%</td>
<td>2.47%</td>
<td>0.66838</td>
<td>1.62%</td>
<td>1.58%</td>
</tr>
<tr>
<td>G3 (Fine)</td>
<td>0.01430</td>
<td>2.36%</td>
<td>2.16%</td>
<td>0.66838</td>
<td>1.69%</td>
<td>1.58%</td>
</tr>
<tr>
<td>G4 (Very Fine)</td>
<td>0.01432</td>
<td>3.05%</td>
<td>2.28%</td>
<td>0.66703</td>
<td>1.05%</td>
<td>1.37%</td>
</tr>
</tbody>
</table>

The results show a good agreement with the experimental data and the assessment of the uncertainty shows that there is a small sensitivity of the drag coefficient to the grid density, however already with the medium mesh (G2) the solution is quite consistent.

4.1.2 Steady cavitating flow

In the steady cavitating condition a steady sheet cavity is expected that covers approximately the 20% of the foil. The time step and the number of inner iterations per time step (I_N) are investigated in the coarse mesh by comparing the pressure distribution, the vapor volume fraction and the time history of the total vapor volume. A grid sensitivity study for the lift and drag coefficient is conducted for this condition as well (Table 4).

Table 4: Drag and Lift coefficient values in cavitating flow for all the grids and the computed uncertainty (Uφ).

<table>
<thead>
<tr>
<th>Grid</th>
<th>CD</th>
<th>Uφ</th>
<th>% Δexp</th>
<th>CL</th>
<th>Uφ</th>
<th>% Δexp</th>
</tr>
</thead>
<tbody>
<tr>
<td>G1 (Coarse)</td>
<td>0.01916</td>
<td>15.67%</td>
<td>0.24%</td>
<td>0.6352</td>
<td>0.24%</td>
<td></td>
</tr>
<tr>
<td>G2 (Medium)</td>
<td>0.01849</td>
<td>6.02%</td>
<td>0.68%</td>
<td>0.6621</td>
<td>0.68%</td>
<td></td>
</tr>
<tr>
<td>G3 (Fine)</td>
<td>0.01852</td>
<td>6.00%</td>
<td>0.51%</td>
<td>0.6638</td>
<td>0.51%</td>
<td></td>
</tr>
<tr>
<td>G4 (Very Fine)</td>
<td>0.01838</td>
<td>3.61%</td>
<td>0.35%</td>
<td>0.6648</td>
<td>0.35%</td>
<td></td>
</tr>
</tbody>
</table>

Table 5: Numerically obtained frequencies from various sources (6 deg angle of incidence and σ=1.0) [12].

<table>
<thead>
<tr>
<th>Author</th>
<th>V_inlet (m/s)</th>
<th>Shedding Frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>Koop [13]</td>
<td>12</td>
<td>≈24</td>
</tr>
<tr>
<td>Sauer [14]</td>
<td>12</td>
<td>≈11</td>
</tr>
<tr>
<td>Schnerr et al. [15]</td>
<td>12</td>
<td>≈11.18 (incompressible)</td>
</tr>
<tr>
<td>Oprea [16]</td>
<td>6</td>
<td>≈9 (compressible)</td>
</tr>
<tr>
<td>Hoekstra &amp; Vaz [17]</td>
<td>6</td>
<td>≈14</td>
</tr>
<tr>
<td>Ziru Li [18]</td>
<td>6</td>
<td>≈15.4</td>
</tr>
</tbody>
</table>

Figure 5: Pressure distribution along the foil for different I_N per time step (σ=1.6).
Figure 6: Vapor volume fraction along the foil for different time steps (σ=1.6).
It is concluded that a time step according to a courant number even higher than one can be sufficient (see Figure 6) and the number of inner iterations does not affect the development of the cavity although the vapor volume might not be fully converged in every time step (for instance despite the fact that using a time step corresponding to courant number 0.75 the total vapor volume is converged after 20 iterations per time step, the results between 5, 20 and 100 inner iterations are identical, Figure 5). The uncertainty assessment is in line with the results in wetted flow showing dependence to the mesh (stronger this time) for the drag coefficient. In addition to that, a slight grid sensitivity regarding the shape at the trailing edge of the sheet cavity is observed (see Figure 8).

4.1.3 Unsteady cavitating flow

An investigation on the impact of the number of inner iterations per time step is conducted first. Using 100 inner iterations and a time step corresponding to Courant number 0.75 the solution shows that a number of 40 inner iterations per time step is sufficient (Figure 9). An unsteady periodic cycle is predicted giving a shedding frequency of about 3.6 Hz (Figure 11). However, according to other numerical studies such a frequency seems to be very low (Table 5). To this end the modification for the turbulent viscosity is applied. The results show that a higher frequency can be achieved with a second order temporal discretization scheme or with lower time step. Eventually a shedding frequency of about 13.6 Hz is computed, using 40 inner iterations, courant number 0.75 and a second order temporal discretization scheme (Figure 11) together with Reboud’s correction for eddy viscosity.

The instantaneous images of the volume fraction are shown in Figure 10. First, as the cloud cavities from the previous cycle are moving downstream, a sheet cavity starts to grow at the leading edge in combination with some cavities growing at the trailing edge (steps 1-3). The re-entrant jet is formed and moves towards the leading edge as the bubbly cloud from the previous cycle collapses (steps 4-5). Then the sheet cavity starts to shed (step 6) and as it becomes smaller and smaller it continues shedding (steps 6-8) till it completely disappears (step 9). Finally all the shedding parts are combined into a bubbly cloud and move downstream to the trailing edge as the sheet cavity starts to grow again at the leading edge and the new cycle starts (steps 10-12).
A grid sensitivity study for the shedding frequency has also been conducted. The results are shown in Table 6. With every mesh a shedding frequency between 13 and 14 Hz has been computed. The high uncertainty of the solution can be explained by the unsteadiness and randomness of the shedding.

**Figure 9:** Convergence of the residuals and the total vapor volume in every time step.

**Figure 10:** Total Vapor volume, drag and lift coefficient during a typical shedding cycle.

**Figure 11:** Total vapor volume in time and frequency domain for NACA 0015 with (right) and without (left) Reboud’s correction for eddy viscosity.
4.2 NACA66 (mod.)-312

The NACA66 hydrofoil as described before is used to validate the computational set-up. The results are compared with experimental data obtained by Leroux et al. for two different conditions. The experimental and the computational obtained data are shown in Table 7.

Table 6: Grid sensitivity study on the shedding frequency and assessment of the uncertainty for each mesh for the NACA 0015.

<table>
<thead>
<tr>
<th>Grid density</th>
<th>Shedding (Hz)</th>
<th>Uφ</th>
</tr>
</thead>
<tbody>
<tr>
<td>G1 (Coarse)</td>
<td>13.35</td>
<td>15.40%</td>
</tr>
<tr>
<td>G2 (Medium)</td>
<td>13.89</td>
<td>22.44%</td>
</tr>
<tr>
<td>G3 (Fine)</td>
<td>13.50</td>
<td>12.32%</td>
</tr>
<tr>
<td>G4 (Very Fine)</td>
<td>13.23</td>
<td>7.39%</td>
</tr>
</tbody>
</table>

Table 7: Experimental frequency and Strouhal number based on chord length as measured by Leroux et al. Vₑ=5.33m/s, Re = 0.8 x 10⁶ [10].

<table>
<thead>
<tr>
<th>Experiment</th>
<th>CFD</th>
</tr>
</thead>
<tbody>
<tr>
<td>α</td>
<td>f (Hz)</td>
</tr>
<tr>
<td>5.5</td>
<td>2.88</td>
</tr>
<tr>
<td>6.0</td>
<td>3.50</td>
</tr>
<tr>
<td>7.0</td>
<td>4.50</td>
</tr>
<tr>
<td>8.0</td>
<td>18.00</td>
</tr>
</tbody>
</table>

Figure 12: Total vapor volume in time and frequency domain for 6 deg (top) and 8 deg (bottom) angle of incidence.

Figure 13: Experimental-numerical comparison on the NACA66 for 6 deg angle of incidence. Experimental images are computed from an average of three instantaneous periods (Δt=1/50 s) and compared with computed instantaneous void fraction images with the same period.

The results show that in both cases a frequency similar to this in the experiment is obtained. There is a difference less than 3% for the low frequency case and 2% for the high frequency case. A comparison between the computations and the experimental data of the foil in 6 deg angle of incidence is shown in Fig. 13. As illustrated by Leroux et al. two steps can be identified during a typical shedding cycle: The first step consists of the growth of the sheet cavity (Figure 13 a-e) till it is slowed down and counterbalanced by the shedding of vapor structures.
Themistoklis I. Melissaris, Norbert Bulten and Tom van Terwisga

(secondary clouds) in the wake (Figure 13 f-i). After the shedding of secondary clouds, the detachment of a large vapor cloud (main cloud) occurs (Figure 13 j). It is followed by the roll-up and convection of the main cloud (Fig. 13 k) together with the growth of the residual cavity. The second step occurs just after the cavity break-off. Indeed, the growth of the residual cavity is abruptly stopped at nearly the same time the main cloud of vapor collapses (Figure 13 l), and the residual cavity almost entirely disappears (Figure 13 m). Then the cavity starts to grow again.

Similar cavitation dynamics are calculated by the simulations. The growth of the cavity and the secondary clouds are captured as well as the detachment of the large vapor cloud and the sudden vanishing of the cavity after the collapse of the cloud. Discrepancies can only be observed on the growth of the residual cavity, where a larger expansion of the residual cavity is predicted in the computations (the same behaviour was also predicted in the computations by Leroux et al).

5 CONCLUSIONS AND RECOMMENDATIONS

In this study an attempt was made to verify the incompressible RANS solver in StarCCM+ in cavitating flow. Despite the three-dimensional character of cavitation dynamics a first investigation was conducted on the grid and numerical (time step, inner iterations etc.) sensitivity with the intention to predict the shedding frequency using two-dimensional domain. For the current computational set-up and the tested conditions the following conclusions are drawn:

- When a steady sheet cavity is predicted the effect of the time step and the number of the inner iteration on the results are negligible. However, the grid density had a slight impact on the shape of the sheet cavity.
- In the unsteady condition, on the other hand, the time step and the number of inner iterations per time step seem to play an important role on the prediction of the shedding frequency. A higher frequency was captured only when the correction for the turbulence viscosity in areas with higher vapor volume was applied. Without the correction the effect of the re-entrant jet could not be captured thoroughly, leading to a “delayed” shedding and consequently to a lower shedding frequency.
- Furthermore, it should be noted that the number of iterations needed per time step changes for different time steps and order of temporal discretization, so it is suggested that they are selected in such a way that convergence of the total vapor volume per time step is achieved.
- A grid independent solution has been reached; even the coarsest mesh was capable of capturing the dynamic shedding in a high frequency after application of Reboud’s eddy viscosity correction.
- As a second step, an effort to validate the model was made comparing the numerical results with experimental data. Good agreement was obtained and the shedding frequency was accurately predicted. Discrepancies can only be observed in the maximum total volume per cycle.
- Further computations on a three-dimensional domain are recommended to investigate possible alterations on the cavitation dynamics and the shedding frequency.
• It is finally recommended to investigate the possible erosion mechanisms and the capability of predicting the potentially erosive cavitation implosions using incompressible URANS solver.

REFERENCES


A VOLUME-PRESERVING INFLOW BOUNDARY BASED NUMERICAL TANK APPLIED TO WAVE-STRUCTURE INTERACTION IN NEAR-SHALLOW WATER

SHASWAT SAINCHER∗† AND JYOTIRMAY BANERJEE†

†Department of Mechanical Engineering, Sardar Vallabhbhai National Institute of Technology
Surat 395007, Gujarat, India. Fax: +91 261 2228394
e-mail∗: shaswat.saincher@gmail.com, web page†: http://www.svnit.ac.in

Key words: Volume-preserving, Navier-Stokes, numerical wave tank, steep waves, kinematic stretching, wave-structure interaction

Abstract. Inflow-boundary based Navier-Stokes (NSE) wave tanks are prone to volume addition, especially whilst generating steep \((H/\lambda > 0.03)\) waves in near-shallow water \((kh < 1)\). In the present work, a volume-preserving, inflow boundary based numerical tank is proposed in the two-phase NSE framework. Volume effluxed under troughs is balanced against that influxed under crests using kinematic stretching. The wave tank is tested for generation of steep \((H/\lambda = 0.037, 0.048)\), trochoidal waves in near-shallow water \((kh \approx 0.8)\). A comparison with baseline inflow formulation demonstrates that the proposed inlet boundary effectively restricts volume addition without inducing wave distortion. The NWT model is later implemented to wave-structure interaction occurring during low frequency \((T = 2\ s)\) wave propagation over a submerged trapezoidal bar. Good agreement with experimental data is reported.

1 INTRODUCTION

Numerical wave tanks (NWTs) are the computational counterpart of wave flumes. Over the past three decades, NWTs have emerged as a much needed secondary standard (to wave flumes) for addressing a wide variety of challenging problems in marine and coastal engineering as well as in physical oceanography. A few important examples include vortex dynamics of breaking waves, performance evaluation of wave energy converters [1], motion response of ships in waves and wave-structure interaction [2]. In order to design numerical algorithms for addressing the above-mentioned problems, one requires prior knowledge of the degree of complexity involved in each case. This would in turn enable one to develop algorithms with an optimum level of complexity that ensures a sustainable utilization of computational resources. For instance, the kinematics of ocean wave propagation is (for the most part) irrotational which lends simplification to numerical modeling and even opens avenues for analytical treatment in some cases. An apt example for the latter would be the development and subsequent application of the quasi-determinism theory [3] towards simulation of freak waves, wave groups and interaction and diffraction of wave groups past obstacles in a semi-analytical framework. Naturally, many of the pioneering (and even some
recent) works in NWT modeling were based on a potential (inviscid) flow assumption using the Boussinesq or Laplace equations [2]. The potential flow approach has an obvious advantage in that the framework of equations governing wave motion has the free surface elevation $\eta$ and velocity potential $\phi$ as explicit variables for which the system could be solved. In addition to this, the $\phi - \eta$ framework allows a (relatively) coarser spatio-temporal discretisation to be realized in the NWT ($\lambda/\Delta x \sim 30$ and $T/\Delta t \sim 40$) [2] without risking numerical damping of the generated waves. Inviscid NWT formulations have been successfully applied to problems such as non-linear wave propagation, wave transformation and generation and evolution of focused waves [2].

Despite its advantages, the inviscid framework is insufficient for describing the hydrodynamics of waves if there is an occurrence of large velocities $\gg C$ (where $C$ is wave celerity) and/or if there is significant interplay between air and water “phases”. Apt examples for this include wave breaking, wave mechanics in steep sea-states, interaction of breaking waves with structures and hydrodynamics of oscillating water column (OWC) type wave energy converters [1]. In such scenarios, the effect of molecular (and/or turbulent) viscosity and/or the presence of the air phase on wave kinematics can no longer be neglected. This necessitates development of viscous NWT models [4, 5] based on the Navier-Stokes equations (NSE) that, in reality, govern ocean wave propagation and facilitate the highest fidelity description of a given problem.

In case of NSE formulations, $\eta$ and $\phi$ no longer (explicitly) appear as variables and the same need to be implicitly linked to the framework of governing equations. In a viscous NWT, the free surface displacement $\eta$ needs to be correlated to the topology of the air-water interface. This necessitates a two-phase description of the fluid domain which could be achieved using (say) the VOF method [6]. Further, the potential $\phi(t)$ (determined apriori from wave theory) can be used to impose local, Dirichlet prescriptions of $U(t), V(t)$ (and $\eta(t)$) at so-called “inflow-boundaries” [4] that act as wave-generating boundary conditions for the momentum equations. However, inclusion of $\phi$ in NSE-based wavemaker formulations is not mandatory as wave generation could simply be achieved by specifying a “mass-source”, based on $\eta(t)$, in the continuity equation for few cells comprising a “source region” [5]. In contrast to inviscid wave tanks, there is added complication in viscous NWTs to minimise artificial damping imposed by various numerical approximations made in the flow solver [6]. The need for minimizing numerical damping imposes limits on the coarsest spatio-temporal resolution that is permissible for a given wave design. Experience with numerical wave generation by the authors [6] indicates that NSE-based NWTs demand very fine temporal resolution ($T/\Delta t \geq 3000$) when $H/\lambda \leq 0.01$ (especially if $kh < 1$) whilst a refined spatial resolution ($\lambda/\Delta x \geq 150$) becomes necessary when $H/\lambda > 0.03$. These numbers are in sharp contrast to those mentioned earlier for a $\phi - \eta$ based NWT model [2].

It is evident that two-phase Navier-Stokes based NWT simulations are challenging. It can be further stated that the task of viscous wave generation becomes especially difficult in near-shallow water ($kh < 1$). The difficulty manifests itself in two ways:

- the water column has poor $U$—momentum damping characteristics for $kh < 1$. Wave generation is hence prone to occurrence of wave-vorticity interactions that (might) induce height damping in near-field of the wavemaker [6] and
- Stokes drift induced by the waves increases as $(\sinh kh)^{-2}$. Therefore, NWT simulations in $kh < 1$ are prone to wave setup resulting from volume addition.
The problem of volume addition can be prevented using a source-function wavemaker which is designed based on free-surface elevation $\eta(t)$ of the target waveform [5]. Since $\int_{T}^{T} \eta(t) \, dt = 0$ over a wave period $T$, source-function based NWTs have excellent volume conservation properties. However, when $kh < 1$, jets of volume ejected from the source region lead to pervasive vortex formation which induces wave distortion and viscous height damping in the near-field [6]. The issue of near-field height damping can be resolved using an inflow-boundary based generator which is designed based on $U, V, \eta(t)$ of the targeted waves. In this case, initial momentum lost to vorticity is minimal because volume is influxed (rather than ejected) into the domain through the inflow-boundary. However, (water) volume influx and efflux into the tank depends on $U(y, t)$ and $\int_{T}^{T} U(y, t) \, dy \, dt \neq 0$ at any order (of Stokes theory). Hence, there is net volume addition through an inflow boundary which induces setup for longer simulations ($t > 10T$).

Apparently, the task of designing a NSE-based NWT for $kh < 1$ is faced with contrasting challenges that necessitate modifications in baseline wavemaker formulation; such a modification has already been proposed by the authors [6] in a source-function based wavemaker for $kh \approx 0.8$. However, if volume-preserving, an inflow-boundary based wavemaker should be preferred over a source-function technique because (a) there is an obvious reduction in the number of design variables and (b) the computational domain is only to be modeled at one end of the wavemaker. Hence, in the present work, we extend the capabilities of our existing NSE-based PLIC-VOF NWT model [6] by introducing a volume-preserving inflow boundary based wavemaker and a simplified methodology for treatment of immersed boundaries. The mathematical formulation of the tank is presented in section 2. Volume conservation properties of the baseline inflow boundary have been improved using the concept of kinematic stretching [9] which is discussed in section 2.3. Boundaries of immersed structures in the tank (if any) have been approximated using non-uniform, stair-stepped meshes employing a staggered variable arrangement; said implementations are presented in section 3. The proposed NWT model has been tested for generation of steep regular waves ($H/\lambda = 0.037, 0.048$) in near-shallow water and validated against the experiments of Beji and Battjes [10] involving sinusoidal low frequency (SL) wave transformation over a submerged trapezoidal bar. Results obtained from these benchmarking tests are discussed in section 4. Major conclusions are presented in section 5.

2 Numerical wave tank

In the present work, an existing PLIC-VOF based NWT model, developed previously by the authors [6] has been modified by replacing the mass-source based wavemaker with an inflow-boundary based formulation. In addition, a simplified methodology (involving stair-stepping of the mesh) is introduced for simulating interactions between waves and submerged structures. Mathematical model of the tank is detailed in the following subsections.

2.1 Governing equations

The numerical wave tank has been modeled in a two-phase NSE framework. Wave propagation is considered as a simultaneous flow of air and water using the VOF method. This is done by defining the volume fraction $f$ as the fraction of the primary phase (water) within a
computational cell. The transport of $f$ is governed by the unsteady advection equation (1),

$$\iiint_{C_\Omega} \frac{\partial f}{\partial t} d\Omega + \iiint_{C_S} \left( \vec{V} f \right) \cdot d\vec{A} = 0$$

(1)

where, $\vec{V}$ denotes the advecting velocity, $t$ is time and $d\vec{A}$ is the area of the surface surrounding the control volume $d\Omega$. Equation (1) is solved geometrically using Youngs PLIC-VOF algorithm which is based on the recurrence of two steps; interface reconstruction and interface advection. In the first stage, the air-water interface is reconstructed using the gradient of the volume fraction field. The advection of the reconstructed fluid region is operator-split and comprises of consecutive sweeps along the $x$ and $y-$ directions. The sweeping direction is alternated every time step for second-order accuracy and a conservative volume redistribution algorithm is run after each sweep to eliminate overshoots ($f > 1$) and undershoots ($f < 0$) in the volume fraction field. Wave motion is governed by solution of the two-phase NSE,

$$\iiint_{C_\Omega} \vec{V} \cdot d\vec{A} = 0$$

$$\iiint_{C_\Omega} \frac{\partial U}{\partial t} d\Omega + \iiint_{C_S} \left( U \vec{V} \right) \cdot d\vec{A} = -\frac{1}{\rho^*} \iiint_{C_\Omega} \frac{\partial p}{\partial x} d\Omega + \frac{1}{\rho^*} \iiint_{C_S} \left( \mu^* \nabla \vec{U} \right) \cdot d\vec{A}$$

(2)

$$\iiint_{C_\Omega} \frac{\partial V}{\partial t} d\Omega + \iiint_{C_S} \left( V \vec{V} \right) \cdot d\vec{A} = -\frac{1}{\rho^*} \iiint_{C_\Omega} \frac{\partial p}{\partial y} d\Omega + \frac{1}{\rho^*} \iiint_{C_S} \left( \mu^* \nabla \vec{V} \right) \cdot d\vec{A} - \iiint_{C_\Omega} g d\Omega$$

where, $U$ and $V$ denote the streamwise ($x$) and vertical ($y$) components of velocity, $p$ is the pressure, $\rho^*$ and $\mu^*$ are the mixture density and mixture viscosity respectively and $g$ is the acceleration due to gravity. The mixture properties are calculated as,

$$\rho^* = f \rho_w + (1-f)\rho_a \quad \text{and} \quad \mu^* = f \mu_w + (1-f)\mu_a$$

(3)

where subscripts $w$ and $a$ denote the water and air phases respectively. As mentioned previously, there is an added task of minimising numerical damping of wave height in NSE-based NWTs [6]. It is difficult to eliminate all sources of numerical damping, however, as suggested by Perić and Abdel-Maksoud [7], the errors introduced by numerical approximations can be reduced (but not eliminated) by designing a sufficiently fine computational mesh. Hence, in NSE-based NWT formulations, more emphasis is laid upon mesh design (especially for steep waves) for arresting artificial damping [5, 6, 7]. NWT domain configuration and meshing strategy are presented in the next subsection.

2.2 Domain and mesh configuration

The computational model of the numerical wave tank is shown in figure 1. Two separate configurations of the tank (differing chiefly in the manner in which waves are damped) have been formulated for simulating regular trochoidal waves and transformation of SL waves. Model parameters have been represented here in a generalized manner; problem-specific values of design variables are reported in section 4. With reference to figure 1, L is the length of the wave
Simulation region, \( H \) is the height of the domain, \( \ell_d \) is the length of sponge layer and \( h \) is the still water depth. Variables characterizing the mesh structure are also shown; \( nx \) and \( ny \) represent the total number of cells along the \( x \) and \( y \) directions respectively, \( nxm \) is the number of cells in the wave simulation region, \( nxr \) denotes the number of cells in the east sponge layer (ESL) and \( nyu, nyd \) denote the number of vertical cells above and below the still water level (SWL) respectively. Two additional variables have been derived: number of cells per wavelength \( nx_\lambda \) and per wave height \( ny_H \). These variables help in correlating the mesh design to the steepness (\( H/\lambda \)) of the targeted waves. This in turn is helpful in establishing general rules of thumb for selecting optimum mesh refinement for different target wave designs [6].

2.3 Numerical wave generator and sponge-layer design

An inflow boundary-based wavemaker has been employed for wave generation in the present NWT which involves a Dirichlet prescription of \( \eta(t), U(y,t) \) and \( V(y,t) \) at a vertical boundary (see figure 1). The generator is based on Stokes V theory [8] which accurately predicts the dynamics of ocean waves traveling in finite depth and deep water. At fifth order, local free surface displacement \( \eta(t) \) is given by,

\[
\eta(t) = k^{-1} \left\{ A \sin(\omega t) - (A^2 B_{22} + A^4 B_{24}) \cos(2\omega t) - (A^3 B_{33} + A^5 B_{35}) \sin(3\omega t) \right\} \\
+ k^{-1} \left\{ A^4 B_{44} \cos(4\omega t) + A^5 B_{35} \sin(5\omega t) \right\}
\]
where, $k$ is circular wavenumber, $A$ is a topological parameter and $\omega$ is the circular frequency. The coefficients $B_{22}, B_{24}$ etc. are fractions of polynomials in $\sinh(kh)$ and $\cosh(kh)$ and represent weights assigned to the component harmonics of the Stokes V wave [8]. It should be noted that $A$ and $k$ are not known a priori and are inter-related through the equation set (5),

$$A = \frac{0.5kH}{1 + A^2B_{33} + A^4(B_{35} + B_{55})} \quad \text{and} \quad k = \frac{4\pi^2/\sqrt{gT^2}}{\tanh(kh)(1 + A^2C_1 + A^4C_2)}$$

where $H$ is the target wave height, $T$ is wave period and the coefficients $C_1$ and $C_2$ govern amplitude dispersion at fifth-order,

$$C_1 = \frac{8c^4 - 8c^2 + 9}{8s^4}$$
$$C_2 = \frac{3840c^{12} - 4096c^{10} - 2592c^8 - 1008c^6 + 5944c^4 - 1830c^2 + 147}{512s^{10}(6c^2 - 1)}$$

where $s \equiv \sinh(kh)$ and $c \equiv \cosh(kh)$. Equation set (5) is solved iteratively for $A$ and $k$ using initial guesses from linear wave theory. The velocities $U, V$ are in turn derived from the potential $\phi(y,t)$ at fifth order,

$$\frac{2\pi T \phi(y,t)}{\lambda^2} = (AA_{11} + A^3A_{13} + A^5A_{15}) \cosh(ky) \cos(\omega t) + (A^2A_{22} + A^4A_{24}) \cosh(2ky) \sin(2\omega t) - (A^3A_{33} + A^5A_{35}) \cosh(3ky) \cos(3\omega t) - A^4A_{44} \cosh(4ky) \sin(4\omega t) + A^5A_{55} \cosh(5ky) \cos(5\omega t)$$

where, the coefficients $A_{11}, A_{13}$ etc. are weights assigned to component harmonics of the Stokes V wave [8]. Equation (6) represents the “baseline formulation” of the inflow technique [4] which is only suitable for wave generation at low Ursell numbers $(Ur)$. With increasing $Ur$, there is a non-negligible addition of mass into the domain. Due to a two-phase nature of the simulation, the added mass induces wave setup which lessens accuracy. In the present work, the issue of mass addition is addressed through a modified inflow formulation that employs kinematic stretching [9]. Here, the velocity potential $\phi$ in equation (6) is altered by replacing coordinate $y$ with a stretched coordinate $\zeta$,

$$\frac{2\pi T \phi(\zeta,t)}{\lambda^2} = (AA_{11} + A^3A_{13} + A^5A_{15}) \cosh(k\zeta) \cos(\omega t) + (A^2A_{22} + A^4A_{24}) \cosh(2k\zeta) \sin(2\omega t) - (A^3A_{33} + A^5A_{35}) \cosh(3k\zeta) \cos(3\omega t) - A^4A_{44} \cosh(4k\zeta) \sin(4\omega t) + A^5A_{55} \cosh(5k\zeta) \cos(5\omega t)$$

such that $\zeta \equiv y \cdot \frac{h + \varphi(t)}{h + \eta(t)}$ where $\varphi > 0$. It is noteworthy that the condition $\varphi = 0$ corresponds to Wheeler’s method [9] which kinematically over-designs the troughs but under-designs the crests. While Wheeler’s method may be suitable for nearly sinusoidal waves at moderate steepness, it significantly under-predicts crest momentum for strongly non-linear waves where the troughs are closer and crests are farther from the SWL. The methodology proposed in equation (7) is novel in that the prescription of $\varphi$ is flexible and depends on the wave design in question; the
induced setup can hence be directly controlled using $\varphi$. The method described above is termed as the modified inflow technique. Trochoidal wave generation using the baseline (equation (6)) and modified (equation (7)) inflow techniques is compared later in section 4.

A wave absorption strategy is essential in NWTs for preventing modulation of the incident wave train by energetic reflections occurring from the far end of the tank. In the present work, incident wave energy has been absorbed/dissipated using a sponge layer for regular wave simulations and a beach for wave-structure interaction simulations. The chief reason for using a beach in the latter case is to closely replicate the experimental conditions of Beji and Battjes [10]. In case of the sponge layer, a combination of momentum damping and grid coarsening [6] has been used for ensuring maximum absorption of incident wave energy. The wave-induced velocity field is damped using equations,

$$\tilde{U} = \tilde{U} - e^{-\alpha(1-x^*)R} U^n \quad \text{and} \quad \tilde{V} = \tilde{V} - e^{-\alpha(1-x^*)R} V^n$$

(8)

where $\tilde{U}, \tilde{V}$ are predicted values of the streamwise and depthward velocity (neglecting pressure), $U^n, V^n$ are the previous time level values, $\alpha (= 10)$ is the strength of the sponge layer, $R (= 1)$ controls the variation of damping along the length of the layer and $x^* = \frac{|x-x_a|}{\ell_d}$ is a normalized $x-$coordinate which is zero at the beginning of the sponge layer ($x = x_a$) and unity at the eastern boundary of the tank ($|x-x_a| = \ell_d$). The sponge layer length ($\ell_d$) has been selected such that $\ell_d > 3\lambda$. It should be noted that $R$ is kept constant in time and equation (8) is applied before correcting the velocity field for continuity. The latter step ensures that the velocity field available for PLIC-VOF advection is divergence-free at the beginning of a new time level. Numerical treatment of submerged structures is described in the next section.

3 Numerical treatment of immersed boundaries

In the present work, immersed boundaries such as bars and beaches have been modeled following an “obstacle approach” (see figure 2). Pressure (or VOF) cell-centers falling inside the outline of the bar/beach are flagged; the flagged cells are skipped during momentum/pressure calculations. The elimination of flagged cells from the domain leads to a characteristic “stair-stepped” approximation to the immersed boundary. It should be noted that no local modifications in cell sizes were introduced for improving said stair-stepped approximations.

The geometrical approximation is followed by local imposition of no-penetration conditions (in $U$ and $V$—momentum) along the entire stair-stepped boundary. A backward-staggered variable arrangement (in addition to maintaining tight pressure-velocity coupling) ensures that $U$ and $V$—momentum cell centers get exactly placed along the stair-stepped approximation thereby greatly simplifying assignment of no-penetration conditions. As observed from figure 2, previously flagged $p, f$ cells are now employed as “ghost cells” (centroid highlighted in yellow) for imposing no-penetration conditions adjacent to fluid cells. It should also be noted that boundary treatment for the beach is identical to that of the bar frontal slope and is not shown here for the sake of brevity.

The proposed methodology thus facilitates a simplified treatment of non-Cartesian geometries in the NWT even when the placement of solution variables is staggered. Wave generation performance of the NWT is evaluated in the next section.
4 Results and discussion

The present NWT model is tested for steep (monochromatic) wave generation in near-shallow water as well as propagation and transformation of SL waves over a submerged trapezoidal bar [10]. The first category of problems is used for demonstrating superior volume conservation properties of the proposed inflow boundary. The second category is chosen for testing fidelity of the immersed boundary treatment strategy described in section 3.

4.1 Steep trochoidal waves

Two trochoidal wave designs have been considered; a steep case \( C \) (\( H = 9 \text{ cm}, H/\lambda = 0.037, U_r = 19.5 \)) [6] and a steeper case \( C_a \) (\( H = 12 \text{ cm}, H/\lambda = 0.048, U_r = 27.2 \)). Both designs are 1.5 s waves propagating in 30 cm deep water (\( kh < 0.8 \)); their topologies being governed by Stokes V theory [8]. Identical NWT setups have been considered for both designs. Referring to figure 1, \( \mathbb{L} = 19.0 \text{ m}, \ell_d = 10.0 \text{ m} \) and \( H = 0.6 \text{ m} \). Corresponding mesh is designed using \( nxm = 1330, nxe = 50 \) (stretched) and \( nyu = nyd = 25 \) (stretched) yielding \( nx_{\lambda} = 170 \) and \( n_{yH} = 6 \). These values are decided following the design criteria previously established by the authors [6]. Time is non-uniformly advanced with Courant number limited to \( C_{\text{max}} \leq 0.25 \) and time step size limited to \( \Delta t_{\text{max}} \leq T/750 \). Pressure field \( (p) \) in the simulation is initialized following the hydrostatic law \( p = \rho g (h - y) \forall y \leq h \). The NWT is run for \( t = 20T \) in each case.

Results of NWT simulation for cases \( C \) and \( C_a \) are shown in figure 3. In each case, local variation of free surface elevation measured +6\( \lambda \) away from the wavemaker is compared for baseline \((\varphi = \eta(t); \zeta \equiv y)\) and modified \((\varphi > 0 \forall y < h; \varphi = \eta(t) \forall y \geq h)\) inflow formulations. A superior performance of the proposed “modified inflow” boundary is clearly evident as setup
induced in wave topology due to volume addition is convincingly nullified (figures 3 (a,b)). The baseline and modified inflow configurations are also compared based on percentage change in primary phase (water) volume ($\mathcal{V}_E$) within the NWT [6] during last five wave periods (figure 3 (c)). It is seen that there is net volume addition over a wave period in the baseline formulation ($\varphi = \eta(t)$) whose magnitude increases with steepness $H/\lambda$. This is also evident from the stronger setup induced for case $C_a$ in figure 3 (b). It naturally follows that the “optimum” value of $\varphi$ required for exactly balancing volume addition would also increase with $H/\lambda$ which is indeed the case from figure 3. It should be noted that $\varphi = 0.265$ and $\varphi = 0.335$ considered for cases C and $C_a$ (respectively) have been determined parametrically (said analysis not shown here).

Results demonstrate that the proposed modified inflow boundary based NWT is volume preserving and that momentum over-design imposed below the SWL does not induce any wave

Figure 3: Trochoidal wave generation performance of the proposed NSE-based NWT model illustrated using $\eta(t)$ signals measured +6$\lambda$ from the wavemaker for cases (a) $C$ and (b) $C_a$ and (c) time variation of volume error ($\mathcal{V}_E$) during last five wave periods of the simulation.
distortion far from the wavemaker. Wave-structure interaction simulations performed using (only) the modified inflow formulation are presented next.

### 4.2 Wave transformation over a submerged trapezoidal bar

Long wave propagation over barred topographies is characterized by extensive short wave generation on the lee side [10] which offers an invaluable strategy for coastal protection. Simulation of wave transformation scenarios poses a unique difficulty in terms of optimizing spatio-temporal resolution in the NWT because $H/\lambda$ and $kh$ are variable along the direction of propagation.

In the present work, it is aimed to replicate SL wave transformation experiments of Beji and Battjes [10]. The experiments involve propagation of small steepness ($H = 2\text{ cm}, H/\lambda = 0.005$), low frequency ($T = 2.0\text{ s}$) waves in $40\text{ cm}$ deep water ($kh = 0.68$) over a submerged bar and the “transformed” wave train is eventually dissipated over a gently sloping beach; geometries

![Figure 4](image)

**Figure 4:** Time series of spatial profiles $\eta(x)$ showing non-breaking transformation of sinusoidal low frequency waves ($H = 0.02\text{ m}, T = 2\text{ s}, h = 0.4\text{ m}$) propagating over a submerged trapezoidal bar for a duration of $t = 11T$. Geometries of the bar and beach are shown at the bottom; coordinates are in m.
of the bar and beach are shown, to scale, in figure 4. Despite the fact that $H/\lambda = 0.005$, $Ur = 4.27$ which renders the waves slightly trochoidal thereby necessitating Stokes V theory for representation. Correlating Beji and Battjes’ setup to the computational model in figure 1, $L = 29.95\, m$, $H = 0.44\, m$ and $h = 0.4\, m$. Mesh design is characterized by $nxm(= nx) = 1500$, $nyu = 20$ and $nyd = 26$ (stretched) yielding $nx/\lambda = 185$ and $ny/H = 10$. Given small steepness of the upstream waves, $\Delta t_{max}$ was (conservatively) set to $T/5000$ (following parametric investigations not shown here) with $C_{max} \leq 0.25$; the limiting value $C_{max}$ was only exceeded during initial breaking at the beach. Pressure field within “fluid cells” (see figure 2) was initialized using the hydrostatic law and the simulation was run for $t = 15T$.

Major stages of the wave transformation process are highlighted in figure 4. The SL waves

Figure 5: Validation of normalized free surface elevation signals measured at six wave gauge locations (WG2–WG7) [10] in the NWT after a passage of nine wave periods for a duration of $t = 4.5T$. 
initially shoal over the front (seaward) face of the bar and become progressively asymmetrical \((t = 4T)\). Triplet resonance occurs over the straight portion of the bar \([10]\) which generates dispersive trailing “free” waves \((t = 6T)\). As the train “de-shoals” \([10]\) behind the bar, the primary wave breaks up into several smaller amplitude waves with nearly harmonic frequencies \((t > 9T)\). The present simulations have also been compared against local free surface elevation \(\eta(t)\) measurements of Beji and Battjes at six wave gauge locations (see figure 5). The comparison is initiated from a downcrossing in each case and, barring minor phase differences at some stations, the overall agreement between PLIC-VOF NWT and experiments is good. The phase difference is (probably) attributable to a forward Euler method being used for time advancement; it is aimed to increase the order of time discretization in the near-future.

5 Conclusion

We propose a volume preserving, wave-inflow technique and a simplified numerical treatment of immersed boundaries for NSE-based NWTs. Said methods have been successfully benchmarked against steep wave generation and wave-structure interaction scenarios for \(kh < 0.8\).

REFERENCES


AN INVESTIGATION ON THE EFFECTS OF TIME INTEGRATION SCHEMES ON WEAKLY COMPRRESSIBLE SPH METHOD

MARINE 2017

DENIZ C. KOLUKISA†, MURAT OZBULUT* AND EMRE PESMAN‡

†,‡Karadeniz Technical University, Faculty of Marine Sciences, Surmene Deniz Bilimleri Fakultesi, Camburnu, 61530 Trabzon, Turkey
e-mail: *dckolukisa@ktu.edu.tr, ‡pesman@ktu.edu.tr, www.ktu.edu.tr

* Piri Reis University, Faculty of Engineering
Piri Reis University Seaside Campus
Postane Mahallesi, Eflatun Sk. No:8, 34940 Istanbul, Turkey
e-mail: mozbulut@pirireis.edu.tr, www.pirireis.edu.tr

Key words: SPH Method, Time Integration Schemes, Free Surface Flows, Dam Break Problem

Abstract. Temporal discretization is a key aspect of the weakly compressible Smoothed Particle Hydrodynamics (SPH) method, as existing studies prove that the time integration schemes affect the stability of the simulations of weakly compressible SPH [1]. In this study, accuracy and performance of the classical 4th order Runge-Kutta method as a time integration scheme was evaluated by comparing simulation results of 2D dam break problem in terms of pressure and free surface profiles with single step (Euler method), predictor-corrector (midpoint) schemes and existing simulation results given in the literature. Density correction algorithm was utilized as a baseline treatment to prevent density fluctuations. The effect of Artificial Particle Displacement (APD) algorithm is another numerical treatment which is investigated in the present work. It is observed that APD provides more homogeneous particle distribution, leading to a higher accuracy. As for the comparison between time integration schemes, results based on the free surface deformation indicate that the Runge-Kutta method achieves success at reducing the free surface particle scattering encountered on Euler and midpoint schemes.

1 INTRODUCTION

Smoothed Particle Hydrodynamics (SPH) method is a meshless numerical method mainly utilized for simulating fluid flow problems. With its fully Lagrangian and meshless nature, the method facilitates simulations of complex geometries and large deformations. Therefore interest of the researchers from ship hydrodynamics studies intensifies with the development and expansion of the method's capabilities.

The SPH method was emerged from astrophysics research by the studies of Gingold and Monaghan [2] and Lucy [3]. Subsequently Monaghan [4] modified the method to simulate free surface fluid flow problems by carrying out dam break, wave maker and beach wave propagation simulations. Monaghan proposed an incompressible fluid approximation by limiting the compressible characteristic of SPH method which was referred to as Weakly
Compressible SPH (WCSPH) approach [4].

In this study, non-viscous Euler equations were implemented, which require an artificial viscosity term in momentum equation to compensate spatial discretization effects [5]. The value of the artificial viscosity term is significant, since it should be determined in conformity with the discretization parameters [6]. In terms of temporal discretization, performance of the classical 4th order Runge-Kutta method as a time integration scheme was compared with implementation of single step Euler method and predictor-corrector midpoint method. The effects of the artificial viscosity were also examined by implementation of four different kinematic viscosity value; 5x10^{-3}, 1x10^{-3}, 5x10^{-4} and 1x10^{-4} [m^2/s]. Simulations of 2D dam break problem was run applying three different spatial discretization setup by representing fluid domain with 7200, 11250 and 16200 particles for each time integration scheme and kinematic viscosity combination. Results of the simulations were compared with the experiment results of Pakozdi [7] and SPH simulation results of Ozbulut et al. [8] to present suggestions for future studies. Furthermore an adaptive artificial particle displacement (APD) algorithm along with velocity variance based free surface (VFS) algorithm was implemented.

2 NUMERICAL MODELING

2.1 Governing Equations

Effects of viscosity can be neglected for the dam break problem. Equation of motion for Newtonian fluids with neglected viscosity is defined by Euler's equation:

$$\frac{du}{dt} = -\frac{1}{\rho} \nabla p + g$$  \hspace{1cm} (1)

$$u = \frac{dr}{dt}$$  \hspace{1cm} (2)

where $u$, $r$ and $g$ are velocity, position and gravitational acceleration vectors; $\rho$ and $p$ are density and pressure respectively. The continuity equation is expressed as follows:

$$\frac{1}{\rho} \frac{dp}{dt} = -\nabla u$$  \hspace{1cm} (3)

In WCSPH method, pressure is determined via implementation of an equation of state for gases. In this study equation of state proposed by Monaghan [6] is utilized:

$$p = \frac{\rho_0 c_0^2}{\gamma} \left[ \left( \frac{\rho}{\rho_0} \right)^\gamma - 1 \right]$$  \hspace{1cm} (4)

where $\rho_0$ is reference density and is equal to 1000 [kg/m^3] for fresh water, $\gamma$ is the ratio of heat for water and is equal to 7 and $c_0$ is the reference speed of sound. Incompressibility is simulated by enforcing density fluctuations under 1% of the reference density value, which is achieved by restricting Mach number ($M$) under 0.1, thus limiting value of $c_0$ [4, 8, 9]. In this study $c_0$ is taken as 50 [m/s].
2.2 SPH Discretization

In SPH, fluid domain is represented by freely moving particles which are treated as interpolation points. As a Lagrangian method, SPH allows these particles to retain their physical identities throughout the simulation period. Value of a function $f$ in a discretized SPH domain is represented with the *kernel approximation*:

$$ f(x) = \int f(x')W(x - x', h) \, dx' $$

(5)

where $x$ and $x'$ represent coordinate sets of two separate points. $W$ is the kernel function which works as a weighting factor depending directly on the distance between given points limited by the definition of smoothing length "$h". Interpolated sum of the neighboring data for an SPH particle is represented with the *particle approximation*. While the neighbor particles of particle $i$ are denoted by $j$, value of the function $f$ and its gradient for particle $i$ in the discretized SPH domain are calculated by following equations:

$$ \langle f_i \rangle = \sum_{j=1}^{n} \frac{m_j}{\rho_j} f_j W_{ij} $$

(6)

$$ \langle \nabla f_i \rangle = \sum_{j=1}^{n} \frac{m_j}{\rho_j} f_j \nabla W_{ij} $$

(7)

where $m$ and $\rho$ represents the mass and density of the particles respectively. Various kernel functions are available in literature for utilization of different purposes [10, 11]. In this study quintic spline kernel function was utilized:

$$ W(R, h) = \alpha_d \begin{cases} 
(3 - R)^5 - 6(2 - R)^5 + 15(1 - R)^5, & 0 \leq R < 1 \\
(3 - R)^5 - 6(2 - R)^5, & 1 \leq R < 2 \\
(3 - R)^5, & 2 \leq R < 3 \\
R \geq 3 
\end{cases} $$

(8)

where $R = r_{ij}/h$, and $r_{ij}$ is distance of the particle $j$ to the particle $i$ in operating dimension; $\alpha_d$ is a coefficient depending on the dimension of the simulation domain, in this case taken as $7/(478\pi h^2)$ for two dimensions.

Numerical discretization of the governing equations of fluid motion is done by the particle approximation on the Euler's equation and continuity equation, explained with detail in [12]:

$$ \frac{d u_i}{dt} = -\sum_{j=1}^{N} m_j \left( \frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2} + \Pi_{ij} \right) \cdot \nabla_i W_{ij} $$

(9)

$$ \frac{d \rho_i}{dt} = \rho_i \sum_{j=1}^{N} m_j \left( \frac{u_i - u_j}{\rho_j} \right) \cdot \nabla_i W_{ij} $$

(10)

where time rate of change of velocity and density of the particle $i$ are calculated directly.
without reducing the total derivatives to their local and convective components since we follow the particle according to the Lagrangian description. Artificial viscosity ($\Pi_{ij}$) term in momentum equation (7) is expressed as:

$$\Pi_{ij} = \begin{cases} -\alpha \mu_{ij} \frac{c_i + c_j}{\rho_i + \rho_j}, & u_{ij} \cdot r_{ij} < 0 \\ 0, & u_{ij} \cdot r_{ij} \geq 0 \end{cases}$$  \hfill (11)

$$\mu_{ij} = \frac{h (u_i - u_j) \cdot (r_i - r_j)}{||r_i - r_j||^2 + \theta h^2}$$  \hfill (12)

$$v = \frac{1}{\delta} \alpha \tau c$$  \hfill (13)

where $c_i$ is the local speed of sound for the particle calculated by $c_i = c_0 (\rho_i / \rho_0)^{(\gamma - 1)/2}$, $\alpha$ is a constant related to kinematic viscosity ($\nu$) value with respect to the Equation (13) recommended by Monaghan and Kos [6] and $\theta$ is a parameter which has a constant value of 0.05.

### 2.3 Correction Algorithms

**Density correction**: Since the pressure calculation depends on the deviation of density from the reference value by $7^{th}$ order (see Equation 4), small density disturbances can lead to high oscillations in pressure field. Therefore, a density correction algorithm [8] as a baseline treatment was applied in simulations:

$$\hat{\rho}_i = \rho_i - \sigma \frac{\sum_{j=1}^{N} (\rho_i - \rho_j) W_{ij}}{\sum_{j=1}^{N} W_{ij}}$$  \hfill (14)

where $\hat{\rho}$ is the corrected density and $\sigma$ is an averaging constant taken as 1 in this work.

**Hybrid VFS+APD algorithm**: Velocity variance based free surface (VFS) and artificial particle displacement (APD) algorithms are corrective tools utilized for preventing particle clustering and noisy pressure. In this study a generalized hybrid formulation was implemented, integrating VFS into APD algorithm covering both free surface and fully populated fluid regions:

$$\sigma u_i = \frac{\sum_{j=1}^{N} (u_i - u_j) W_{ij}}{\sum_{j=1}^{N} W_{ij}}$$  \hfill (15)

$$\sigma r_i = \sum_{j=1}^{N} \frac{r_{ij}^2 u_{eff} W_{ij}}{r_{0}^2}$$  \hfill (16)

$$\hat{u}_i = u_i - \epsilon \sigma u_i, \quad \hat{r}_i = r_i - \sigma r_i$$  \hfill (17)

where $\hat{u}_i$ and $\hat{r}_i$ are the corrected velocity and position vectors, $\epsilon$ is a constant parameter taken as 0.003, $r_0$ is an averaged sum of neighboring particle distances calculated for each
particle as \( r_0 = \sum_i r_{ij} / N \), and \( u_{eff} \) is the velocity based APD coefficient linking VFS to APD algorithm calculated by \( u_{eff} = |\sigma u_i| \).

### 2.4 Time Integration Schemes

The general form of the time integration schemes utilized in this work is as follows:

\[
\beta_i^{n+1} = \beta_i^n + \phi \Delta t
\]

While \( \beta_i \) represents particle positions, densities or velocities; the function \( \phi \) varies with the scheme requiring calculation of the derivatives \( du_i/dt \) and \( dp_i/dt \) at the beginning point of the iteration"n" for Euler method, and at the half time step "\( (n + 1/2) \)" for the modified Euler method also known as the midpoint method. The classical fourth order Runge-Kutta method has a more sophisticated determination of function \( \phi \), assessing and weighting the derivatives calculated once at the beginning, twice at the half step and once at the end of the time step.

### 3 NUMERICAL RESULTS

The 2D dam break problem setup is shown in Figure 1. Dimensions of the problem geometry were set with respect to the experiment setup of Pakozdi [7], defined as \( H_w=0.6 \) [m], \( L_w=1.2 \) [m], \( L=3.23 \) [m]. Located on the opposite wall \( H_p=0.115 \) [m] above the ground, pressure measurement point was symbolized with P. Moreover, initial particle pressures for all simulations were set to the Hydrostatic pressure condition.

![Figure 1: Dam break problem setup](image)

Wall boundaries were represented with a single row of solid particles. In order to assure impermeability and for kernel symmetry of the fluid particles near the boundaries, ghost particles were utilized and recreated dynamically at each time step. Particles in the \( \Delta r \leq 3h \) neighborhood of solid boundaries were mirrored with respect to the tangent of the wall. Ghost particles have the same mass and density values with the corresponding fluid particles. However the velocity vectors for the particles were mirrored with respect to the free slip condition to reflect the force equilibrium. In addition, the pressure values of the ghost particles below the horizontal wall were adjusted to reflect the hydrostatic pressure difference with respect to the vertical distance. During the simulations, fluid particles with less than 25 neighbor particles were considered as free surface particles and their densities, thereby pressures were set to reference values.
Smoothing length was set fixed as $1.33\Delta x$ for each simulation, where $\Delta x$ represents the initial particle distances in each axis. In terms of spatial discretization, three cases with different initial particle distances were implemented, representing the same fluid domain with 7200, 11250 and 16200 particles. Simulations were run with fixed time steps. Time step sizes were determined with respect to the CFL condition; $\Delta t \leq C_{\text{CFL}} \frac{h_{ij\text{min}}}{c_i+\nu_{\text{max}}}$ [8]. The $C_{\text{CFL}}$ number was taken as 0.2 for the simulations.

For each time integration scheme and spatial discretization combination, kinematic viscosity values of $5\times10^{-3}$, $1\times10^{-3}$, $1\times10^{-4}$ and $1\times10^{-4}$ [m²/s] were employed to assess the effects of artificial viscosity. Kinematic viscosity values were switched via alteration of the variable $\alpha$ (see Equation 11) with respect to the Equation 13. Furthermore hybrid VFS+APD algorithm was applied in extra simulations of $\nu=5\times10^{-3}$ to represent the effects. Therefore, a $4\times3\times3$ matrix of 36 numerical simulations were created, given in Table 1.

<table>
<thead>
<tr>
<th>Kinematic Viscosity [m²/s]</th>
<th>Number of Particles</th>
<th>Integration Scheme</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Euler</td>
<td>Midpoint</td>
</tr>
<tr>
<td>$5\times10^{-3}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7200</td>
<td>e1</td>
<td>m1</td>
</tr>
<tr>
<td>11250</td>
<td>e2</td>
<td>m2</td>
</tr>
<tr>
<td>16200</td>
<td>e3</td>
<td>m3</td>
</tr>
<tr>
<td>$1\times10^{-3}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7200</td>
<td>e4</td>
<td>m4</td>
</tr>
<tr>
<td>11250</td>
<td>e5</td>
<td>m5</td>
</tr>
<tr>
<td>16200</td>
<td>e6</td>
<td>m6</td>
</tr>
<tr>
<td>$5\times10^{-4}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7200</td>
<td>e7</td>
<td>m7</td>
</tr>
<tr>
<td>11250</td>
<td>e8</td>
<td>m8</td>
</tr>
<tr>
<td>16200</td>
<td>e9</td>
<td>m9</td>
</tr>
<tr>
<td>$1\times10^{-4}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7200</td>
<td>e10</td>
<td>m10</td>
</tr>
<tr>
<td>11250</td>
<td>e11</td>
<td>m11</td>
</tr>
<tr>
<td>16200</td>
<td>e12</td>
<td>m12</td>
</tr>
</tbody>
</table>

Three additional cases were also created utilizing hybrid APD+VFS algorithm on cases m1, m2 and m3. Numerical simulations of the cases were performed by the developed serial C++ code which allows solution of 1 to 14 time steps per second on a single CPU core, depending on the integration scheme and number of particles. In figures 2 to 4; numerical and experimental data of pressure values on the opposite wall at location P were compared in terms of time integration schemes. Results with the same kinematic viscosity values and same number of particles are grouped while only integration schemes vary in each figure.

Figure 2 demonstrate that the viscosity value of $5\times10^{-3}$ [m²/s] does not return satisfying results. Besides the delayed arrival of the fluid to the opposite wall, it can be seen that the pressure values fall short of the experimental data. Obviously, lower kinematic viscosity values should return better results since the kinematic viscosity of water is around $1\times10^{-6}$ [m²/s]. However as mentioned in earlier sections, discretization is a limiting factor for the viscosity value. Utilization of real viscosity value is inconvenient since it requires extremely small smoothing length, resulting in huge number of particles and infinitesimal time stepping; since decreasing $\alpha$ value itself to a limit without decreasing $h$ value provokes instability.
problems causing simulation to collapse.

With a decrease of viscosity, numerical results at Figure 2 d, e and f indicate the accurate impact time and adequately matches the pressure levels with a drop in latter stages. However noise of the pressure values are increased comparing to higher viscosity simulations; in contrast, the noise levels are reduced with the increase of the number of particles.

Noise grows larger with the decrease of viscosity in Figure 3 a, b and c. Besides, the noise reduction with the increase of number of particles becomes clearer. Further at Figure 3 d, e and f, noisy characteristic continues however the pressure drop is avoided at the latter stages. It can be claimed that the time integration schemes have a minor effect on the obtained impact pressure results.

**Figure 2:** Pressure evaluations; a)7200, b)11250, c)16200 particles for $\nu=5\times10^{-3}$ [m$^2$/s]; and d)7200, e)11250, f)16200 particles for $\nu=1\times10^{-3}$ [m$^2$/s]
As a final remark, it can be inferred from Figure 3 that particle discretization should be finer at lower viscosity simulations to reduce the noise levels of pressure. To enhance the accuracy of pressure time series for low viscosity simulations, hybrid APD+VFS algorithm is utilized on cases m1, m2 and m3 and results are displayed in Figure 4. Addition of APD and VFS algorithms into the numerical scheme increases the pressure values significantly and yields more compatible results with the experiment data. Nevertheless, the issue of delay due to low viscosity at the arrival on the opposite wall remains unsolved.
In addition to the pressure results, free surface profiles for m1 with and without APD+VFS algorithms were compared with the existing simulation results of Ozbulut et al. [8] (Fig. 5 and Fig. 6).

**Figure 5**: Free surface comparisons for simulations for m1 at $t = 2.23(H_w/g)^{0.5}$. Left: m1. Right: m1 with APD+VFS

**Figure 6**: Free surface comparisons for simulations for m1 at $t = 5.34(H_w/g)^{0.5}$. Left: m1. Right: m1 with APD+VFS
It should be noted that utilization of APD and VFS with a combined fashion is a factor that alters the results which can be seen from right figure at Figure 6. Local determination of the variable $u_{eff}$ in the present study is the main reason behind this difference.

Figure 7: Free surface comparisons for simulations for $\nu=1 \times 10^{-4}$ [m$^2$/s] at $t=5.34(H_{w}/g)^{0.5}$; with 7200 particles (left), with 16200 particles (right).

In Figure 7, effects of the time integration schemes were compared in terms of free surface scattering. Based on the results from Figures 2 and 3, it is seen appropriate to compare the results of simulations with viscosity values of $1 \times 10^{-4}$ [m$^2$/s], since pressure oscillations are the highest. As it can be seen from Figure 7, Runge-Kutta time integration scheme has a positive effect on reducing the scattered particles due to the impact on the wall.

4 CONCLUSIONS

The results of the present work indicate that lower kinematic viscosity produces more accurate, yet oscillatory pressure time series. In contrast, finer discretization reduces the oscillatory characteristics. At the lowest viscosity value ($1 \times 10^{-4}$ [m$^2$/s]) finer discretization (in this case more than 16200 particles) is required to obtain more satisfying results. On the other hand, at the highest viscosity value ($5 \times 10^{-3}$ [m$^2$/s]) without APD and VFS algorithm, simulations fail to represent the physical phenomena accurately. In fully populated fluid domain, particle positions are corrected with APD algorithm to prevent particle clustering and disorder. And for the free surface, especially at the impact zone, VFS algorithm helps keeping particles together by velocity correction. Hybrid implementation of both APD and VFS gives
the advantage of having easier adaptation of the code to different physical problem cases by providing local evaluation for velocity and position corrections.

Utilization of different time integration schemes has slight differences on pressure results. Since general pressure characteristics of the WCSPH solution of dam break problem is oscillatory, the nuance can be observed from the free surface forms. The classical fourth order Runge-Kutta method gives the slight edge, however simulation times are about 4 times longer comparing to the Euler time integration method or nearly 3 times longer than the midpoint method. The disadvantage should be compensated by parallel computation possibilities.

REFERENCES


COMPARATIVE ANALYSIS OF TIP VORTEX FLOW USING RANS AND LES  
MARINE 2017  
ABOLFAZL ASNAGHI, RICKARD E. BENSO** AND URBAN SVENNBERG†  

**Department of Shipping and Marine Technology  
Chalmers University of Technology, Gothenburg, Sweden  
e-mail: {abolfazl.asnaghi, rickard.bensow}@chalmers.se  
†Rolls Royce AB, Kristinehamn, Sweden  
e-mail: urban.svennberg@rolls-royce.com  

Key words: Tip Vortex, Numerical Simulation, RANS, LES, Cavitation Inception  

Abstract. The current study focuses on the numerical analysis of tip vortex flows, with the emphasis on the investigation of turbulence modelling effects on tip vortex prediction. The analysis includes comparison of RANS and LES methods at two different mesh resolutions. Implicit LES, ILES, modelling is employed here to mimic the turbulent viscosity. In RANS, the two equation k-ω SST model is adopted. In order to also address possible benefits of using streamline curvature variations in RANS, two curvature correction methods proposed for k-ω SST are tested, and compared. ILES results show very good agreement with the experimental observations. The predicted vortex in ILES is also stronger than RANS predictions. ILES has predicted accelerated vortex core axial velocity very well, while tested RANS models under predict the axial velocity. Adoption of curvature correction has not improved the tip vortex prediction, even though it has reduced the turbulent viscosity at the vortex core.  

1 INTRODUCTION  

As the flow passes over a lifting wing with finite span, close to the wing tip the pressure differential drives the fluid from the high pressure side on the lower surface to the low pressure side on the upper surface. This creates a highly rotational vortex flow pattern at the tip region. As this vortex is transported downstream more flow from the wake of the wing is fed into the vortex. This roll-up process will strengthen the vortex until its circulation is nominally equal to that of the wing. The roll-up process typically extends to a few wing spans downstream of the trailing edge, which is denoted the near field region. After this region and when the roll-up is finished, the vortex will start to decay due to the flow viscosity [1,2].  

As the pressure at the vortex core is lower than the surrounding area, in the cavitating case, cavitation incepts at the vortex core. Understanding the physics of these flows, and its modelling, is therefore important in finding the tip vortex inception speed to prevent or control the occurrence of cavitation on propellers [3,4].  

Experimental tests on propellers can provide very useful information about the vortex properties and the tip vortex cavitation inception. However, despite the huge cost which has to be spent for each test, as the tip vortex involves very small scale flow dynamics it is very
difficult to measure relevant flow features, e.g. velocity distribution, even by advanced measurements tools. Another drawback is the disability to measure the pressure at the vortex core where cavitation inception occurs. Numerical tools and CFD can be used to give further insights on the tip vortex properties that experimental tests may not be able to provide [5-7].

The current study focuses on the evaluation of different turbulence modelling approaches in prediction of tip vortex flows around an elliptical foil. The aim is to compare the k-\omega SST RANS model and Implicit LES, which have been used widely in marine applications [8-11]. In the authors’ previous study, mesh resolution requirements for tip vortex analysis in laminar and ILES methods have been conducted [2]. Here, the turbulence models are tested on the optimum spatial resolution. One coarser resolution is also considered to briefly address the impacts of turbulence modelling on different spatial resolutions.

Another objective of the paper is to compare two different curvature correction methods, and how they change the flow prediction in different spatial resolutions. It has been reported that linear eddy viscosity assumption is insensitive to the streamline curvature [12-14]. This leads to over-prediction of the turbulent viscosity in high swirling regions, e.g. the vortex core. As a result, the numerically predicted vortex decays much too fast. Curvature corrections are proposed, essentially for linear eddy viscosity based models such as k-\omega SST, to cure over-prediction of the turbulent viscosity.

The simulations and analysis are conducted on the tip vortex flow on an elliptical foil, namely Arndt elliptical foil. The vortex structures around this type of foil resembles the propeller tip vortex behaviour while making it possible to be tested in more details both experimentally and numerically. The tip vortex at the selected operating conditions is relatively stationary which reduces the computational requirements [15, 16].

The investigations include the comparisons of vortex trajectory, vortex axial velocity at the vortex core, velocity distributions at different in-plane sections downstream of the foil, and the pressure distribution. Turbulent viscosity for different RANS models are also presented to evaluate the effects of curvature correction. Results indicate that ILES is capable of prediction of tip vortex characteristics very accurately in the near field region; accelerated vortex core axial velocity and in-plane velocity distribution matches quite well with experimental PIV images. The SST models, however, fail in correctly predicting the vortex properties. Even though the vortex properties of the RANS simulations, close to the tip, are similar to ILES, the predicted vortex fades very rapidly in RANS.

2 ELLIPTICAL FOIL

The current study focuses on the numerical investigation of the wetted flow of the Arndt elliptical foil. The geometry is an elliptical planform having the NACA 662−145 as cross section having mean line equal to a = 0.8 [1,15-18]. In the current study, the experimental study conducted in the test tunnel of the Laboratory for Ship Hydrodynamics at Delft University is selected for comparison with the numerical results [15, 16]. In the experimental tests, Stereoscopic PIV measurements were employed to provide the velocity distributions at different sections downstream of the foil. Correlation averaging was utilized in the post processing of PIV images in order to minimize the interrogation area size. The foil was tested at different flow conditions; here, the wetted flow conditions with fixed inlet velocity (6.8 m/s) is employed while the foil has angle of attack equal to 9 degrees.
3 GOVERNING EQUATIONS

3.1 RANS Model

The conservation of momentum in RANS framework is,

\[ \frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu + \mu_t \frac{\partial u_i}{\partial x_j} \right]. \]  

(1)

In the current study, k-ω SST model is used to model the turbulent viscosity. In k-ω SST, two transport equations for turbulent kinetic energy and dissipation terms are solved [12],

\[ \frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_j} = \nu \frac{\partial}{\partial x_j} \left[ \mu + \sigma_{k} \mu_t \frac{\partial k}{\partial x_j} \right] \]

(2)

\[ \frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho u_i \omega)}{\partial x_j} = \nu \frac{\partial}{\partial x_j} \left[ \mu + \sigma_{\omega} \mu_t \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \frac{\rho u_k}{\omega} \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_k}. \]

(3)

\[ P = \tau_{ij} \frac{\partial u_i}{\partial x_j}. \]

(4)

\[ \tau_{ij} = \mu_t \left( 2S_{ij} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) \cdot \frac{2}{3} \rho u_i \delta_{ij}. \]

(5)

\[ F_1 = \tanh \left( arg_1^1 \right), \quad arg_1 = \min \left[ \max \left( 2 \sqrt{\beta \omega \left( \frac{500}{\omega} \right)} \right), \frac{\rho \sigma_{kw}}{\sigma_{kk}} \right]. \]

(7)

\[ F_2 = \tanh \left( arg_2^2 \right), \quad arg_2 = \max \left( 2 \sqrt{\beta \omega \left( \frac{500}{\omega} \right)} \right). \]

(8)

The constants are:

\[ \gamma_1 = \frac{\beta_1 \sigma_{kk} \kappa}{\sqrt{\rho}}, \quad \gamma_2 = \frac{\beta_2 \sigma_{kk} \kappa}{\sqrt{\rho}}, \quad \kappa = 2 \sigma_{kk} = 1.0, \sigma_{\omega} = 0.5, \sigma_{\omega} = 0.856, \beta_1 = 0.075, \]

(9)

\[ \beta_2 = 0.0828, \beta^* = 0.09, \alpha_1 = 0.31, \kappa = 0.41. \]

3.2 Curvature Correction

According to the Boussinesq eddy viscosity assumption, a linear relation between turbulent viscosity and the velocity strain rate is considered. This assumption is insensitive to the flow streamline curvature, thus, for highly swirling flows, this can lead to over prediction of turbulent viscosity in the swirling region. Various curvature corrections have been proposed to correct RANS models to also include the curvature variations [12-14].

3.2.1 Menter SST Two-Equation Model with Rotation/Curvature Correction (SST-RC) [12]

This curvature correction form of the SST model is the same as the standard version of SST, except that the production term P in both equations (i.e. transport of k and \( \omega \)) gets multiplied by a function \( f_{r1} \),

\[ f_{r1} = \max\left[ \min\left( f_{rotation}, 1.25 \right), 0 \right], \]

(10)
where
\[ f_{\text{rotation}} = (1 + C_{r1}) \frac{2r^*}{r^*} [1 - C_{r3} \tan^{-1}(r^*)]^{-1} C_{r1}. \]  

All the variables and their derivatives are defined with respect to the reference frame of the calculation, which may be rotating with rotation rate \( \Omega_{\text{rot}} \). Remaining functions are defined as,
\[ r^* = \frac{S}{W} \]  
\[ \hat{p} = \frac{2W_{ln}S_{ln}((S_{ij} + S_{in}S_{jn})\Omega_{\text{rot}}^2)}{WD} \]  
\[ S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \]  
\[ W_{ij} = \frac{1}{2} \left( \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + 2\xi_{mij}\Omega_{\text{rot}}^2 \right) \]  
\[ S^2 = 2S_{ij}S_{ij}, \]  
\[ W^2 = 2W_{ij}W_{ij}, \]  
\[ D^2 = \max(S^2, 0.09W^2). \]  
\[ C_{r1} = 1.0, \quad C_{r2} = 2.0, \quad C_{r3} = 1.0. \]  

The term \( DS_{ij}/Dt \) represents the components of the material derivative of the strain rate tensor. The rotation rate, \( \Omega_{\text{rot}} \), is nonzero only if the reference frame itself is rotating.

### 3.2.2 Hellsten's Simplified Rotation/Curvature Correction (SST-RC-Hellsten) [13, 14]

In this simplified rotation/curvature form of the SST model, the destruction term in the \( \omega \) equation, \( \beta \rho \omega^2 \), gets multiplied by the function \( F_4 \), and becomes, \( F_4 \beta \rho \omega^2 \). The definition of \( F_4 \) is,
\[ F_4 = \frac{1}{1 + C_{RC}R_i}, \]  
where
\[ R_i = \frac{W}{S} \left( \frac{W}{S} - 1 \right). \]  
and \( C_{RC} \) is a constant equal to 1.4.

### 3.3 LES Model: Implicit LES

The filtered Navier-Stokes equation in LES framework is as follows,
\[ \frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \bar{S}_{ij} - B_{ij} \right). \]  

In this equation, \( B_{ij} = \rho_m (\bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j) \) is the subgrid stress tensor and \( \bar{S}_{ij} = \mu (\partial u_i / \partial x_j - \partial u_j / \partial x_i) \) is the shear stress. In ILES model used in this study, no explicit function is applied for B; instead the numerical dissipation is considered enough to mimic the action of B [8-10].
4 COMPUTATIONAL DOMAIN

The computational domain employed here follows the dimensions of the cavitation tunnel at TU Delft. A fixed inlet velocity boundary and fixed outlet pressure boundary is used along with the slip condition on the other boundaries, Figure 1. The inlet velocity is set equal to 6.8 m/s which corresponds to the Reynolds number of 8.5e05. The outlet pressure is set fixed equivalent to cavitation number 4.2. For RANS computations, the inlet turbulence intensity is considered to be equal to 5%.

The inlet and outlet are located five chord length upstream, and ten chord length downstream of the foil. The foil is placed at the center of the channel width, where distance to each side is equal to 150 mm. The root chord length of the foil is equal to 125.6 mm. The center of the coordinate system is located at the center of the chord at the root, and as a result, all of the numerical results in this study are reported accordingly. The trailing edge has been cut off with a thickness of 0.3 mm, and the total area of the foil from the 3D CAD model is 0.01465 m² which is used as the reference area to compute non-dimensional parameters, e.g. lift coefficient. This corresponds to the situation at the experiments.

The tip vortex flows involve very small scales of the flow, both in time and space. Capturing these small scales of flow physics requires very fine computational resolution. The current study focuses on a condition that results in a relatively stationary tip vortex. This reduces the computational time required for the tip vortex to develop and evolve. However, as the vortex diameter is 1 mm, still very fine spatial resolution is required to predict the tip vortex and the flow surrounding it. In previous studies conducted by the authors, a mesh independency study was performed to find appropriate mesh resolution to predict the tip vortex in the near field region, i.e. 1.5 chord lengths downstream of the tip. This distance is deemed appropriate as the main concern of the research is to investigate the tip vortex cavitation inception and its relation with the tip vortex characteristics.

In Figures 2 and 3, general distribution of the cells in the streamwise and in-plane directions are presented. The computational cells are fully hexahedral, generated by employing StarCCM+ of CD-Adapco. Different refinement regions are defined to refine the resolution up to the tip vortex region.

In Table 1, specifications of two mesh resolutions are presented. P1S1 is the coarsest resolution tested in previous studies, and P2S2Wake is the resolution found to be adequate to predict the tip vortex in the near field region [2]. In one of the previous studies, the flow is treated as a laminar flow, and in the other one, ILES was employed. As the tip vortex core has a diameter equal to 1 mm, the sizes in this table are presented accordingly.
Abolfazl Asnaghi, Rickard E. Bensow, and Urban Svennberg

It should be noted that in both of the meshes, the foil surface resolution and prismatic layers’ distribution (and also \( y^+ = 1 \)) are the same, and the only difference between them is the resolution around the tip vortex trajectory.

**Table 1:** Cell size and mesh specifications

<table>
<thead>
<tr>
<th>Name</th>
<th>Total number of Cells (M)</th>
<th>In-plane cell size (mm)</th>
<th>Streamwise cell size (mm)</th>
<th>Number of cells in vortex core</th>
</tr>
</thead>
<tbody>
<tr>
<td>P1S1</td>
<td>8.3</td>
<td>0.125</td>
<td>0.25</td>
<td>8</td>
</tr>
<tr>
<td>P2S2Wake</td>
<td>44.3</td>
<td>0.062</td>
<td>0.125</td>
<td>16</td>
</tr>
</tbody>
</table>

5 SOLUTION PROCEDURE

In order to solve the governing equations, the OpenFOAM package, an open source code written in C++, is used to model and simulate fluid dynamics and continuum mechanics is employed [10].

For the current study, the velocity values on the faces of the computational cells are computed by second order upwinding schemes, `linearUpwind`. For other terms, e.g., turbulent kinematic energy and specific dissipation terms, second order linear scheme is employed. For unsteady simulations, a second order implicit scheme is used for time discretization. The time step is set fixed and small enough to ensure a maximum Courant number to be less than one everywhere in the computational domain for the period of collecting the results. All of the gradients have been corrected to include non-orthogonality effects. The pressure distribution in the vortex core region is presented by using the cavitation index,

$$
\sigma = \frac{p_{\text{int}}}{\rho U_{ref}^2}
$$

6 RESULTS

6.1 Vortex Properties

In Figures 4 and 5, tip vortex properties as predicted by different turbulence models for different spatial resolutions are presented. The comparison includes the variation of cavitation index, vortex trajectory, normalized axial velocity and turbulent kinematic viscosity. To determine the vortex core, a minimum pressure criterion is employed.
In both of the resolutions, and for all turbulence models tested here, negative pressure and also accelerated axial velocity is observed close to the foil, $z/C<0.2$. However, the difference between vortex roll-up and transportation is considerable which contributes to the vortex properties for $z/C>0.2$. In Figure 4, results of P1S1 simulations are presented. The results show over prediction of turbulent viscosity in RANS which damps the vortex after certain distance downstream of the foil. Figure 4(b) indicates that the tip vortex dissipates faster with standard SST compared to using other models. It is noted that after $z/C>0.75$, the vortex continuity breaks and oscillations on the vortex trajectory appears. For SST-RC, the oscillations start after $z/C>1.1$. This correlates to Figure 4(d) which represents the turbulent viscosity. As the highest turbulent viscosity is related to SST and then SST-RC, these models predict faster decay of vortex than SST-RC-Hellsten, and therefore sooner appearance of oscillations in vortex trajectory which relates to vortex breakdown. It is also clear from Figures 4(a) and 4(c) that the pressure and axial velocity in the vortex core quickly dissipates in all RANS models.

Figure 4: Variation of vortex core properties for different turbulence models, P1S1 resolution
When the resolution is increased, in ILES both the vortex strength and its size are increased. In RANS, the strength is also increased causing lower pressure at vortex core close to the tip but the size of the vortex is smaller, and vortex damps out faster, Figure 5(a). The vortex trajectory has less oscillations in the higher resolution, showing more continuous vortex pattern, Figure 5(b). Vortex core velocity predictions, Figure 5(c), show that all of the models predict the accelerated axial velocity, but in RANS as the vortex disappears, the core velocity also reduces. Similar trend in turbulent viscosity are observed for different curvature correction methods, Figure 5(d).

Thus, we conclude that none of the tested RANS models are able to predict the tip vortex correctly in the near field region, while ILES is better in predicting and transporting the vortex downstream in the region of interest, $z/C<1.6$. 

**Figure 5**: Variation of vortex core properties for different turbulence models, P2S2Wake resolution
6.2 Effects of Curvature Corrections

In Figure 6, the distributions of turbulent kinematic viscosity for different RANS models along with \( f_{r1} \) function of SST-RC model are presented at \( z/C=0.5 \) section. As it can be seen from the figure, the SST model is insensitive to the presence of the vortex, which leads to prediction of high turbulent viscosity at the vortex core region. The curvature correction models reduce the viscosity prediction at the vortex core. The comparison between RC and RC-Hellsten indicates that RC-Hellsten predicts lower viscosity at the vortex core region. It also indicates that RC-Hellsten leads to results which are sensitive to mesh variations.

The distribution of \( f_{r1} \) shows that in both resolutions the value of the function is very small, and close to zero in the vortex core region. It indicates that the curvature correction is acting correctly in lowering the production terms in the region where the vortex has evolved. Therefore, other parameters should be involved that lead to inaccurate prediction of the tip vortex. One of the parameters could be the contribution of boundary layer prediction.

![Turbulent Kinematic Viscosity](image)

**Figure 6**: RANS turbulent viscosity, and RC function distributions at \( z/C=0.5 \)

In Figure 7, velocity streamlines of SST-RC and ILES for P2S2Wake resolutions are presented. In frames (a) and (b) of the figure, the close up view of tip vortex are presented; the foil is colored with \( C_p \) distribution. The iso-surface of \( p=p_{sat} \) is also presented with white shaded color to indicate the region with possible cavitation inception. In Figure 7(c), the isometric view of SST-RC is presented along with identifying the zoomed region. For this part, the streamlines are colored with turbulent viscosity.

Comparing ILES and SST-RC results, Figures 7(a) and (b), shows that in ILES the region with lower pressure is larger than SST-RC even though that SST-RC seems to have more concentrated negative \( C_p \) close to the tip; the vortex also starts closer to the tip and slightly further back in the SST-RC simulation. At the bottom of the low pressure region in ILES, Figure 7(a), a high pressure region exists which forces the flow towards the tip vortex. This pressure field sucks more flow from upstream and pressure side to the surface of the foil.
suction side and then towards the tip vortex which as a result leads to a stronger tip vortex prediction in ILES. One uncertainty here is the contribution of low-Reynolds RANS simulation on the boundary layer and therefore pressure field prediction. As it is mentioned in the mesh description section, the spatial resolution employed here have normal resolution y<1. The study to investigate other normal resolutions in RANS simulations are also conducted. The primary results show no improvement in RANS tip vortex prediction. However, further investigations is required in order to make a more precise conclusion.

![Flow streamlines for P2S2Wake resolution](image)

**Figure 7:** Flow streamlines for P2S2Wake resolution

### 6.3 Velocity Distributions

In Figures 8 and 9, the velocity distribution, at z/C=0.5 for P2S2Wake resolution are presented and compared with experiments. Comparison of normalized axial velocity comparison, Figure 8, show that only ILES could predict the accelerated axial velocity while the RANS methods employed here under predict the axial velocity at the vortex core. Under prediction is slightly higher for curvature correction methods.

![Velocity Distributions](image)

**Figure 8:** Comparison between RANS, ILES and experimental normalized axial velocity distributions, P2S2Wake resolution, z/C=0.5
In Figure 9, normalized in-plane velocity distributions are presented. Similar to the axial velocity comparison, ILES provides a more accurate prediction and matches better with the experimental PIV image. The RANS models fail in predicting the peak of the in-plane velocity which relates to the disability to transport the tip vortex until the z/C=0.5 section.

![Figure 9: Comparison between RANS, ILES and experimental normalized in-plane velocity distributions, P2S2Wake resolution at z/C=0.5](image)

7 CONCLUSIONS

Tip vortex simulations of k-ω SST model and ILES are presented in this paper. For SST model, two curvature correction methods are also tested. To include the effects of the resolution on the tip vortex predictions, the simulations are carried out on two different spatial resolutions.

ILES results show very good accuracy in prediction of vortex properties, e.g. axial and inplane velocity distributions. Tested RANS models fail in prediction of accelerated axial velocity. The vortex is much weaker in RANS, which is related to low flow suction into the vortex at the foil tip, and also the contribution of turbulent viscosity over-prediction. Figure 6 shows that the curvature correction methods reduce the turbulent viscosity at the vortex region. However, they could not improve the SST tip vortex predictions, as the results are also related to the flow properties, e.g. foil surface pressure distribution and boundary layer properties. Flow streamlines, Figure 7, show the contribution of suction side pressure field on the vortex formation. For SST-RC-Hellsten model the sensitivity of the turbulent viscosity prediction to the mesh resolution is higher than other RANS models tested. For future steps, it is suggested to test the effects of foil normal resolution, y⁺, on the vortex formation and also interactions of the curvature correction methods with variation of y⁺ for RANS.

8 ACKNOWLEDGEMENT

Financial support for this work has been provided by Rolls-Royce Marine through the University Technology Centre in Computational Hydrodynamics hosted at the Department of Shipping and Marine Technology at Chalmers University of Technology. The simulations were performed on resources at Chalmers Centre for Computational Science and Engineering (C3SE) provided by the Swedish National Infrastructure for Computing (SNIC).

REFERENCES


DISCRETE ELEMENT METHOD SIMULATION OF A SPLIT HOPPER DREDGER DISCHARGING PROCESS

Josip Basic*, Dario Ban*, Nastia Degiuli† and Nicolin Govender‡

* Faculty of Electrical Engineering, Mechanical Engineering and Naval Architecture
University of Split, Rudera Boskovica 32, Split, Croatia
e-mail: jobasic@fesb.hr, dario.ban@fesb.hr - Web page: www.fesb.hr

† Faculty of Mechanical Engineering and Naval Architecture
University of Zagreb, Ivana Lucica 5, Zagreb, Croatia
e-mail: nastia.degiuli@fsb.hr - Web page: www.fsb.hr

‡ Advanced Mathematical Modeling CSIR
Meiring Naude Road, Pretoria, South Africa
e-mail: govender.nicolin@gmail.com - Web page: www.csir.co.za

Key words: Discrete Element Method, Radial Basis Function, Polynomial RBF, Ship Stability

Abstract. Split Trailing Suction Hopper Dredgers (Split TSHD) have longitudinally-split hull, which symmetrically opens when executing gravity-driven unloading of the cargo, while being exposed to various environmental conditions. Even though they have variable hull geometry, their hydrostatic and stability characteristics are usually calculated for initial and unchanged loading conditions only, which is a requirement imposed by classification society stability regulations for TSHD ships [2, 3, 4]. In order to investigate the significance of the discharge process dynamics on actual ship stability, unsteady numerical simulations were performed with the Discrete Element Method (DEM) for symmetrical hopper opening during cargo discharge procedure, without the hull opening failure modes examined. The ship hydrostatic properties, which are pre-calculated analytically using Radial Basis Functions (RBF) for all possible states [11], are used in combination with the solver in order to compute the righting moment and the righting arm, which are affected by the dynamics of the cargo and the loss of displacement. The dynamics of the cargo discharge process was simulated with a DEM solver implemented for Graphics Processing Units (GPUs), Blaze-DEMGPU [8]. Spherical shapes of particulate elements were employed to model the soil cargo, with both cohesion and buoyancy effects included for wetted elements. The simulations of the discharging were performed for various loading conditions. Numerical simulations indicate that the dynamics of the cargo during its discharging should not be ignored due to its effect on the transverse stability of the ship. Therefore, an incoming wave and other environmental loads in combination with a hull opening failure during the discharge could lead to inapt unstable situation of the ship. Non-symmetrical Split TSHD ship openings will be examined in future work, with an investigation of its influence on ship stability and safety of cargo discharge procedures in failure modes.
1 INTRODUCTION

Trailing Suction Hopper Dredgers (TSHDs) excavate large amounts of material from the sea floor and they transport the material to another place. A special type of a dredger ship called Split Trailing Suction Hopper Dredger (Split TSHD) is built with longitudinally-split hull, which opens to perform gravity-driven cargo discharging process, as depicted in Figure 1. The main advantage of the Split TSHD is the fast cargo discharge on a certain near shore position, using mass flow from the silo-shaped cargo hold. Split TSHDs have both sides connected by hinges and the opening is controlled with hydraulic cylinders. While discharging the cargo, failure of the main electric power supply and/or the main hydraulic unit and/or single failure of the normal control systems can occur. Furthermore, as the two halves of split hopper hull are only connected with the hinges and cylinders, a single failure of each of these elements could lead to a catastrophic failure [1]. These opening failures can produce non-symmetric ship geometry, with high risk of reduced ship stability that should be assessed. Moreover, initially trapped water or water that leaped above the freeboard negatively impacts the stability particulars, as it is located at a relatively high position (increasing the ship’s centre of gravity and decreasing the initial metacentric height) and may be associated with an additional free surface moment [1]. The shifting of the cargo can have major effect on the stability of a ship, and usually it is not taken into account for the stability and seakeeping calculations, but the ship construction and its cargo are considered as one rigid body. On the other hand, sandy and gravel cargoes that TSHDs are carrying are not constrained to the ship construction and are submissive to rapid movements. A schematic view of the mass flow in the process of emptying the hopper, while the ship draught is rapidly reduced is shown in Figure 2. The cargo shape, a hull opening malfunction and external loads (wind, waves, trapped water, etc.) can cause force concentration of one side of the hopper and reduce the ship stability in this process. Usual hydrostatic and stability calculations

![Figure 1: Split TSHD main frame section, with split hulls in a discharge condition rotated around the joint](image)

---

---
are conducted for initial, non-variable hull geometries. Current stability regulations regarding Split TSHDs [2, 3, 4] cover initial loading conditions thus avoiding intermediate ship positions and belonging hydrostatic particulars calculations. However, full numerical calculations must overcome complexities of defining the variable ship geometry for the range of opening angles, and calculation of belonging hydrostatic particulars for the each case. Efficient solutions for these ship geometry variations are analytical geometry description methods. An analytical solution of the ship geometry description based on polynomial RBFs (PRBFs) [5, 6] is used in this study. The calculation of hydrostatic calculations for arbitrary heel angles is enabled based on [7], which enables hull affine geometry transformations using rotation, needed for the description of a opened Split TSHD. Moreover, all theoretical ship hydrostatic particulars are pre-calculated in advance for all possible loading conditions, containing three degrees of freedom for a quasi-static ship condition of the draught, the angle of trim and the heel angle [11].

In order to investigate the significance of the discharge process dynamics on actual ship stability, Discrete Element Method (DEM) unsteady numerical simulations were performed for hopper opening during the cargo discharge procedure, without the hull opening failure modes examined. The dynamics of the cargo discharge process was simulated with a DEM solver implemented for Graphics Processing Units (GPU), Blaze-DEM [8, 12]. Spherical elements were used to model the soil cargo, with both cohesion and buoyancy effects included for the wetted elements. Simulations of the cargo discharging were performed for various loading conditions. Ship hydrostatic properties are pre-calculated analytically for all possible states and coupled with the solver to compute ship motions caused by the dynamics of the cargo and the loss of displacement. The paper is organized as follows. The numerical methodology divided into three segments is presented in Section 2. The simulated problem is described in Section 3. The results are presented and discussed in Section 4. Finally, conclusions are drawn in Section 5.

2 NUMERICAL METHODOLOGY

The numerical calculation methodology is divided into three segments. Firstly, theoretical ship hydrostatic particulars are precalculated with RBFs in advance, for all possible states. Secondly, the DEM solver is used to fill the hopper with cargo, and to simulate the cargo unloading while opening the longitudinally split hopper. The simulations are done for various heeling scenarios.
And thirdly, unsteady loads from the discrete elements on the hull geometry are analysed in time, in order to find the worst case of undesirable angular momentum for the simulated case.

2.1 Discrete Element Method

The linear momentum of a particle \( i \) is given by:

\[
L_i = m_i v_i
\]

where \( m_i \) and \( v_i \) are the mass and the velocity of a particle \( i \), respectively. The angular momentum of a particle \( i \) is given by:

\[
H_i = I_i \omega_i
\]

where \( I_i \) is the inertia tensor, and \( \omega_i \) is the angular velocity of the particle. Given that all forces \( F_i \) and torques \( T_i \) act on \( i \)-th particle, the problem is reduced to the integration of the change in the linear momentum:

\[
\dot{L}_i = m_i \frac{d^2 x_i}{dt^2} = F_i
\]

and the integration of the change in the angular momentum for an axis fixed to the body:

\[
\dot{H}_i = I_i \frac{d\omega_i}{dt} + \omega_i \times H_i = T_i
\]

where \( x_i \) is the position of \( i \)-th particle. A linear and an angular position of a particle, and its velocity and acceleration are obtained after the integration. Forces \( F_i \) consist of the surface traction and body forces, and torques \( T_i \) are calculated based on the surface traction forces. The surface traction on a particle \( i \) are the result of contacts with other particles, contacts with boundaries of the domain or applied external loads. An assumption in many DEM simulations is that particles are considered to be rigid for the duration of collision contacts, since the deformations occur on a time scale which is much smaller than what is required for capturing the macroscopic behaviour of a system. The interaction forces between particles \( i \) and \( j \) in contact is resolved through constitutive relationship of a linear repulsive force, and a linear dissipative force model, given by:

\[
f_{ij} = k \delta_{ij} + \gamma_0 \dot{\delta}_{ij}
\]

where \( \delta_{ij} \) is the contact distance, \( \dot{\delta}_{ij} \) is the velocity with which the contact distance changes, \( k \) (N/m) is the normal contact stiffness, and \( \gamma_0 \) is the normal contact damping. The friction resistance as a result of sliding due to the relative tangential surface velocity \( v_{ij}^f \) between particles \( i \) and \( j \) in contact is resolved as:

\[
v_{ij}^f = v_{ij} - n_{ij} (n_{ij} \cdot v_{ij})
\]

which depends on the relative surface velocity \( v_{ij} \) between particles and the contact normal \( n_{ij} \). The relative surface velocity at the point of contact is due to the relative translation and rotation of the two particles:

\[
v_{ij} = v_i - v_j + r_i \times \omega_i + r_j \times \omega_j
\]
where $r_i$ is the vector from particle centre of mass to the contact point. The relative tangential surface velocity $v_{ij}^t$ in addition to Coulomb’s law dictates the tangential force with direction such that it opposes the motion of the two particles. A drawback of the DEM is the large computational cost of collision detection, which scales with the number of particles and geometric complexity. Therefore, particle shapes are mostly approximated with a sphere, although polyhedral shaped particles can capture real particle shape without introducing non physical artefacts.

In this study, Blaze-DEM solver [8] is employed for the dynamic simulation of hull opening and the cargo discharge process. Blaze-DEM is a modular GPU based discrete element method (DEM) framework that supports polyhedral shaped particles by employing a ray-tracing type methodology. On a modern GPU (nVidia GTX 980 Ti), one time step take of a DEM solver takes around ten milliseconds for one million of particles. Usual time step needed for the simulation to be stable is around $\Delta t < 10^{-3}$ s.

![Figure 3: Schematic view of cargo pressure acting on the hopper](image)

### 2.2 Hydrostatic Particulars Using RBFs

It is shown in [5] that 2D ship geometry with discontinuities can be described using the composition of cubic and linear PRBFs, thus enabling further solutions of basic ship hydrostatic integrals, necessary for further stability calculations [7]. The main advantage of PRBFs over other description methods is the possibility of an analytical description of a curve with discontinuities. The composition of linear and cubic PRBFs with dense distribution of points around discontinuities is exact solution of one-dimensional discontinuity description problem with

$$
 f(x) = \sum_i w_i |x - t_i|^3 + \sum_l w_l |x - t_l| + c
$$

(8)

where $w$ is the weight of a point, $t$ is the point position vector, $i$ is the point in the input data set, $l$ is the point in the set of discontinuity data set, $c$ is the shape parameter. After the geometry is globally described using the composition of a cubic and a linear polynomial RBF, hydrostatic particulars of a semi-immersed body can be calculated directly, together with
belonging particulars for arbitrary orientation of the body. Complete hydrostatic particulars, for the outer as well as for the inner ship geometry, can be calculated for the specified range of drafts and heel angles. Interested readers are referred to [7, 11] for the complete explanation of the methodology.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Length, overall, $L_{OA}$</td>
<td>98.0 m</td>
</tr>
<tr>
<td>Breadth, $B$</td>
<td>18.5 m</td>
</tr>
<tr>
<td>Draught, max, $T$</td>
<td>6.55 m</td>
</tr>
<tr>
<td>Hopper capacity, $V_h$</td>
<td>4700 m$^3$</td>
</tr>
<tr>
<td>Hopper length, $L_h$</td>
<td>52 m</td>
</tr>
<tr>
<td>Deadweight, $DWT$</td>
<td>6 000 t</td>
</tr>
<tr>
<td>Speed, loaded, $v$</td>
<td>12 knots</td>
</tr>
<tr>
<td>Propulsion engines power, $P_B$</td>
<td>2 x 3300 kW</td>
</tr>
<tr>
<td>Hull opening angle, $\alpha$</td>
<td>13°, for each side</td>
</tr>
<tr>
<td>Hull opening time, $t_\alpha$</td>
<td>~ 45 s</td>
</tr>
</tbody>
</table>

Table 1: Main ship particulars

### 2.3 Cargo Dynamics and Transverse Stability

The forces that discrete elements impose on the hopper tank are summed for each time-step while the simulation is being performed, as shown in Figure 3. One-way quasi-static coupling is considered, such that for each DEM simulation step, hydrostatic particulars are analysed, i.e. read for the pre-calculated database. The value of the dynamic righting arm is obtained as:

$$GZ(t) = \frac{RM(t)}{F(t)},$$

where $RM$ is the righting moment, and $F$ is the resultant force vector calculated by summing all relevant forces of the free body. Total mass and its centroid is obtained for each time instant as a combination of time-invariant mass of the ship without the cargo, and the mass of the cargo left in the hopper. The immersed volume centroid and buoyancy force value for the current heel angle of the ship is obtained directly by reading pre-calculated database explained in Section 2.2. Therefore, the righting arm for a specific time instant $GZ(t)$ can be calculated from Eq. (9), based on the ship weight, the cargo weight and dynamic forces, buoyancy forces, and the calculated mass and the centre of mass. If it is considered that the change of the shape of the ship in the vicinity of the waterline does not happen abruptly, metacentric height for small angles of heel can be expressed in the following way:

$$GM(t) = GZ(t) \sin \varphi.$$  (10)
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Gravels</th>
<th>Sands</th>
</tr>
</thead>
<tbody>
<tr>
<td>Specific gravity</td>
<td>2.5 - 2.8</td>
<td>2.6 - 2.7</td>
</tr>
<tr>
<td>Bulk density (t/m$^3$)</td>
<td>1.45 - 2.30</td>
<td>1.40 - 2.15</td>
</tr>
<tr>
<td>Dry density (t/m$^3$)</td>
<td>1.40 - 2.10</td>
<td>1.35 - 1.90</td>
</tr>
<tr>
<td>Void ratio</td>
<td>0.25 - 1.0</td>
<td>0.30 - 0.54</td>
</tr>
<tr>
<td>Angle of internal friction</td>
<td>35 - 45</td>
<td>32 - 42</td>
</tr>
<tr>
<td>Permeability (m/s)</td>
<td>&gt; 0.01</td>
<td>$10^{-7}$ - $10^{-3}$</td>
</tr>
</tbody>
</table>

Table 2: Indicative values of soil properties for cohesionless gravels and sands

3 THE PROBLEM DESCRIPTION

3.1 The Ship

The main characteristic of a Split TSHD that makes that kind of ship different than other ship types, is that her geometry is not constant during the unloading process. Her outer and/or inner compartments change in time; inner compartments except cargo hold remain with constant geometry. The ship modelled in this study has three inner compartments on the main frame section: one symmetric cargo hold and two equal ballast tanks on each side of the ship, symmetrically and transversely positioned about the centreline plane. Main particulars of the ship considered in this study are given in Table 1. The filling of ballast water in the discharge process is ignored, since the hull opening time is relatively short compared to the pumps flow rate.

3.2 The Cargo

While describing the size and the shape of grains in granular soils, only predominant size and shape ranges need to be mentioned. To include in the description the full range of every particle present is unhelpful as the size and shape descriptors quickly become so broad as to be meaningless (e.g. nearly all sand becomes “fine to coarse” and nearly all gravel becomes “angular to rounded”) [9]. Soil properties can vary over a range but they do have typical values. Table 2 provides examples of values for gravels and sands soil properties, based on [10], which are used as solver input parameters. Of course, properties can be outside the indicated ranges, but the table is indicative of typical ranges. Typical particle size distributions for the marine dredging process taken from [9] are shown in Figure 4. Specimen #4 is used as the simulation input, which is described as silty very sandy gravel, where gravel is fine to medium and sand is medium to coarse. Even though the Blaze-DEM solver used for the dynamic simulation offers novel and fast method for large-scale polyhedral elements described in [13], spherical elements were employed for the discretisation of the cargo, due to their efficiency and high number of performed runs. Each simulation is firstly initialized by a velocity inlet rectangle, positioned above the hopper tank. Discrete elements of various sizes are randomly spawned, based on the distribution shown in Figure 4, and discharged through the inlet to gravitationally fill the compartment. After the cargo movement has stabilized, hull is being opened, force from the mass flow is analysed by summing forces on the compartment, and stability calculations are performed. An example of filled hopper with randomly distributed particle types, ready to be discharged, is show in Figure 5. For the presented case, hopper is filled with around 2 million particles that have total mass of
Figure 4: Particle size distributions and specimen descriptions [9]

Figure 5: Gravitationally filled cargo compartment, 6000 tons of medium gravel and coarse sand, represented with with 2 million particles

6000 t. The particles are coloured by their diameter, from largest to smallest: red, green, blue and yellow. For the most realistic modelling of the cargo, 10+ millions of particles are needed. For the approximation, smallest particles that do not contribute greatly with their mass can be ignored, i.e. substituted by adding some more of the larger particles to compensate for the mass lost by ignoring the smallest particles.
4 RESULTS AND DISCUSSION

Blaze-DEM solver is validated against hopper flow experiments for both polyhedra and spherical particles [12], and an excellent agreement is found in both the flow-rate and pattern of the particles. For non-cohesive particles without the effects of buoyancy included, the fully loaded ship described in Section 4.1 discharges the cargo in about 25 s. With the mentioned effects included, the discharge procedure takes up to 35 s, which is slightly less than reported by ship specifications. The discrepancy can mostly be attributed to the usage of spherical elements instead of more realistic polyhedrals. Two time instants of the discharging process are rendered in Figure 6, with coloured velocities of the particles. Rolling motion of the ship up to heel angle of

![Figure 6: Two time instants of the gravitational hopper discharge](image)

![Figure 7: Hopper section of 1 m length, for the heel angle of 20°](image)
20° did not yield significant difference in overall time needed for the cargo to discharge. Therefore, it can be said that the ship displacement at specific time instant is similar for various simulations of the specific loading case, i.e. for initially set displacement $\nabla (t = 0)$. Therefore, in order to simplify the numerical procedure and to perform inapt loading scenarios, each simulation was executed for the ship for target heel angle where the cargo would slightly tilt and move its centre of mass, in order to analyse righting arm that is dynamically changing in time. The section cut of a simulation is shown in Figure 7. Ship is allowed to roll around the target heel angle, due to non-symmetrical transverse weight distribution along with unsteady forces from the particles acting on hull. Generally, the centre of mass is dynamically changing, and the resultant force and the moment of force is calculated by summing the ship, cargo and buoyancy components, which are used to derive the value of the righting arm for each discrete time instant, as explained in Section 2.3. The righting arm values calculated with Eq. (9) are presented in Figure 8, in the classic sense as if the remaining cargo at a time instant $t$ represents a loading case. Obviously, a critical scenario occurs when the cargo in fully loaded hopper tilts with the ship heel angle, and thus moves centre of mass away from the centreplane. The righting arm reaches negative

![Figure 8: Dynamic righting arm values for various mass of cargo remaining in the hopper](image)

value at 8° of the heel angle, but the loads needed for that heel angle to occur are out of scope of this study. In addition, granular cargo cohesion and inertia help that the cargo tilting does not happen instantly. It can be deduced that relatively impulsive cargo shifting, or imbalanced loading from the start for the fully loaded case has negative impact on the ship stability. The ship can recover its stability by moving the buoyancy centroid with further opening the hopper and by decreasing the draught with the discharging of the cargo. The metacentric height is straightforwardly calculated with Eq. (10) and plotted in Figure 9. Since the ship rolling is allowed around the target heel angle, the obtained roll angle of the ship (around the target heel angles) is shown in Figure 10.
Figure 9: The metacentric height $GM$ values for various mass of cargo remaining in the hopper

Figure 10: The angle of total force of cargo applied on the hull, compared to the heel angle

5 CONCLUSIONS

Problems that can be modelled and solved with the DEM, can efficiently be simulated on modern hardware by employing data-parallel algorithms. Blaze-DEM solver used in this study can simulate over 10 millions of spherical or polyhedral particles on one GPU device, thus enabling relatively fast simulations of cohesive and non-cohesive granular soils used in maritime dredging processes, e.g. silt, sand and gravel and cobbles.

Split TSHD ship that excavates and carry such materials, symmetrically opens up its longitudinally split hull to perform gravity-driven cargo discharging process. This mass flow in a silo-shaped cargo hold is eligible to simulate with DEM. Even though such ships are exposed to various environmental conditions while opening, their hydrostatic and stability characteristics are usually calculated for initial and unchanged loading conditions only. In this work, unsteady DEM solver was coupled with RBF ship stability solver in order to investigate the significance of the Split TSHD discharge process dynamics on its stability.
Simulations were performed for various heel angles of the ship, where the granular cargo would shift to the heeling side, and hopper would open with defined angular speed. Resultant force and moment of force were calculated by summing the ship, cargo and buoyancy components, which were used to derive the value of righting arm for each discrete time instant. The results showed that relatively impulsive cargo shifting, or imbalanced loading from the start for the fully loaded case has negative impact on the ship stability. Such TSHD ship recovers its stability by further opening the hopper (moving the buoyancy centroid) and discharging cargo (decreasing the draught).

Albeit these kind of ships operate mostly near shore, environmental conditions can be harsh. This raises a question whether a disaster can occur for non-symmetrical hopper section caused by hull opening failure in combination with an incoming wave and other loads (e.g. wind, free-surface effects), which will be examined in future work.

REFERENCES


EXPERIMENTAL INVESTIGATION OF THE AERODYNAMIC FLOW IN THE AIRCRAFT CARRIER SKI-JUMP BY MEANS OF PIV

MARINE 2017

R. Bardera *, M. León-Calero† and A. A. Rodríguez-Sevillano††

* Instituto Nacional de Técnica Aeroespacial (INTA)
Ctra. Ajalvir, km 4.5
Torrejón de Ardoz, 28850 Madrid, Spain
Email: barderar@inta.es, web page: http://www.inta.es

† Instituto Nacional de Técnica Aeroespacial (INTA)
Ctra. Ajalvir, km 4.5
Torrejón de Ardoz, 28850 Madrid, Spain
Email: leoncm@inta.es, web page: http://www.inta.es

†† Escuela Técnica Superior de Ingeniería Aeronáutica y del Espacio
Universidad Politécnica de Madrid (UPM)
Plaza del Cardenal Cisneros, 28040 Madrid, Spain
Email: angel.rodriguez.sevillano@upm.es, web page: http://www.upm.es

Keywords: Experimental aerodynamics, PIV, CFD validation, ski-jump ramp

Abstract. Computational Fluid Dynamics (CFD) methods have opened a new field to perform aerodynamic studies saving money and time. The difficulties presented by this method to calculate complex flow field problems imply that CFD validation is needed to provide correct results. Experimental data have recently been used to validate the accuracy of CFD predictions. Particle Image Velocimetry (PIV) has shown to be a powerful tool in the investigation of complex flows. The aim of this paper is to present results from PIV experiments that would be interesting for CFD validation.

Regarding aircraft operations, the short runway available implies the necessity of equipment which helps to take-off performances. Ski-jump ramp system improves aircraft performances by an increment of lift resulting in successful take-off operations. The ski-jump ramp presence generates a complex flow bounded by a turbulent shear layer and a low velocity recirculation bubble over the end of the flight deck. The adverse effects on the aircraft aerodynamics affect to pilot safe operations, so this region is an interesting problem to be studied by means of wind tunnel experimental tests.
1 INTRODUCTION

Aircraft carriers are essential for military operations, providing tactical advantages to support maritime tasks far from their country [1, 2]. Despite routine operations, pilots have to deal with challenging environments [3, 4]. The structure of aircraft carriers is designed with practical rather than aerodynamic considerations [5]. Moreover, the superstructure is essentially a combination of bluff bodies [2, 6] with sharp edges and corners, antennas, stacks, radomes… that generate a very complex flow field [4, 7]. Hence, pilots must fly through a highly turbulent flow structure [2].

Flow field configuration is affected by wind direction, ship motion and aircraft operational attitude [1, 2]. Regarding that helicopter operations can be performed at different conditions [8], the flow encountered by the helicopter near or over the flight deck is highly complex and unsteady [9, 10]. Crosswind direction is the critical one due to the fact that the massive flow detachment generates a turbulent shear layer and a low velocity recirculation bubble over the flight deck [1, 2, 11]. Consequently, pilots have to handle with workload increment [4, 12] and low safety levels during on board operations [13].

Regarding airplane take-off manoeuvre, the short runway available implies the necessity of equipment which helps to operate in these conditions. Ski-jump ramp system improves aircraft performances by an increment of lift resulting in successful take-off operations [14]. Ahead wind condition is the common configuration during take-off. The ski-jump ramp presence at the end of the flight deck changes the flow structure, so aircraft have to deal with non-stationary aerodynamic effects caused by a turbulent shear layer which delimits the recirculation bubble region [11].

The study of this problem has become more common by using wind tunnel tests [15] and computational methods [16]. Previous researches developed at INTA showed the adverse aerodynamic effect generated in this ship model at ahead wind conditions in the central section of the ski-jump ramp [15].

Advances in Computational Fluid Dynamics (CFD) methods have opened a new field to perform aerodynamic studies that save money and time. The difficulties presented by this technique to calculate complex flows mean that CFD validation is needed in order to provide correct results [13, 17, 18]. Experimental data are continuously used to validate the accuracy of CFD predictions [7, 19].

In the last decades, Particle Image Velocimetry (PIV) has shown to be a powerful tool in the investigation of complex flows [20] to obtain insight about flow during wind tunnel experimental simulation. In addition, PIV shows to be a useful technique for CFD validation [21].

In the present paper, the flow field in the vicinity of a ski-jump ramp has been investigated by testing a 1:100 scaled model of an aircraft carrier. The test experiments were performed in the large wind tunnel INTA-T1 of Instituto Nacional de Técnica Aeroespacial (INTA). PIV technique was used to measure the flow field velocity in the region of interest. The aim of this paper is to present PIV results that play a fundamental role to validate numerical simulations by means of CFD numerical codes.
2 WIND TUNNEL SET UP

2.1 Wind tunnel

The experiments were carried out in a low-speed wind tunnel at the facilities of Instituto Nacional de Técnica Aeroespacial (INTA), located in Madrid (Spain). The wind tunnel INTA-T1 has a closed-circuit and an open test section of $2 \times 3$ m$^2$. The power supply system consists on a DC 450 kW electric motor at 420 V, which provides a maximum freestream velocity of 60 m/s and a turbulence intensity lower than 0.5%.

Figure 1 shows the aircraft carrier model located in the test section oriented with an incident wind angle of 0°. The ship model is located over a platform that simulates the sea surface. The interference between the platform and the incident flow field is reduced by using streamlined leading and trailing edges [22].

Regarding sharp-edged structures, experimental test conditions for a Reynolds number higher than the critical Reynolds number (typically $1.1 \times 10^4$) ensures similarity between the model and the full-scale aircraft carrier [23]. Hence, this condition is fulfilled for a wind tunnel velocity of 10 m/s at ahead wind direction for a Reynolds number of $1.3 \times 10^5$ based on the height of the ramp.

![Figure 1: Ship model located in wind tunnel test section](image)

2.2 Ship model

The aircraft carrier under study is a LHD (Landing Helicopter Dock) ship type which includes a ski-jump ramp to facilitate take-off operations [14] and a superstructure where bridge, radomes, antennas, stacks and other systems are installed [4, 7]. The ship model was made of wood and built at a 1:100 geometric scale to avoid wind tunnel blockage effects. As the platform simulates the sea surface, the model only represents the ship structure above the waterline whose dimensions are indicated in Figure 2.

The non-symmetry of the ship means that airplanes would deal with 3D effects. Hence, three sections along the spanwise of the ski-jump ramp were studied to provide information to be applied in CFD validation process.
Figure 1: Aircraft carrier model dimensions (in millimetres).

Figure 2: Aircraft carrier model dimensions (in millimetres).

3 PARTICLE IMAGE VELOCIMETRY

Particle image velocimetry (PIV) is a non-intrusive technique that measures fluid flow providing instantaneous velocity fields. The high accuracy [20], the ability to obtain quantitative information and the capability to make global velocity measurements have elevated PIV to a special status in fluid mechanics [24]. Regarding the good results that PIV provides for turbulent flow studies [20], the flow structure due to the ski-jump ramp presence was analysed using this technique.

PIV equipment is constituted basically by the following components [24]: a system to generate particles with the required characteristics to seed the flow under study; a light source with enough energy to illuminate the region of interest; a CCD (charge-coupled device) camera to capture the light scattered by those particles; a synchronizer to control the trigger of all devices; and a software to process the images and compute the velocity vectors from the tracer particles displacements. Notice that, the laser system needs a cooler equipment and a set of lens to generate the laser light sheet. Figure 3 shows the PIV equipment used for these experiments.

Figure 3: PIV equipment

As we can observe in Figure 1, the open test-section of the wind tunnel allows to capture the flow information about the region of interest. A Laskin nozzle atomizer [25] was used to
obtain olive oil tracer particles with a diameter of the order of 1 µm [26]. The characteristics of these tracer particles allow to scatter enough light and to avoid buoyancy and inertial problems. In addition, the seeding system is located downwash the test section, so the particles follow the closed-circuit and have enough time to be adapted to the flow velocity conditions.

A neodymium-doped yttrium aluminum garnet (Nd:YAG) laser was used to illuminate the particles transported by the flow. Two 190 mJ laser pulses were delivered, with a time separation (Δt) of 25 µs between them, to compute the velocity components by avoiding blurred particle images. The data rate acquisition of PIV images was 10 Hz (10 pairs of images per second).

The images were recorded by a high-resolution CCD camera (2048 × 2048 pixels) that captures the light scattered by the tracer particles and an AF Nikkor 80-200 camera lens. Notice that, the light laser sheet is located parallel the CCD sensor to avoid misalignment problems.

The synchronizer is in-charge to control the laser pulse and the camera exposure times ensuring that the camera records the flow information at the moment in which the laser is illuminating the flow. This parameter selection is introduced in the computer located in the control room where the different systems are connected.

The dimensions of the captured images are based on the camera field of view (FOV). For this experiment, a FOV of 250 × 250 mm² was obtained by calibrating the FOV with a standardized mesh located inside the laser light sheet. With the aim to fully capture upstream flow information and recirculation bubble region, two images (A and B) were assembled providing one map at each section (Figure 4). The reference axes were selected taking into account the Field of View (FOV) dimensions and region of interest. The image assembly was developed using an in-house Matlab code. The overlapped region reduces slightly the total dimensions of the image. In addition, three sections of the ski-jump ramp were studied to provide 3D information. The non-symmetry of the structure implies that the selected regions to be studied are S₁ (25 %), S₂ (50 %) and S₃ (75 %) of the ski-jump ramp spanwise (see Figure 4).

![Figure 4: Ski-jump ramp dimensions and sections studied.](image-url)
The velocity vector maps were obtained by using two sequential captures in the same area with a time separation ($\Delta t$) of 25 µs. These images are subsampled via an interrogation window scheme. The selected size of the interrogation window was $32 \times 32$ pixels [24] with a 50% overlapping, following the Nyquist sampling criteria [27], as Figure 5 shows. The light scattered by the tracer particles is received by the CCD sensor that provides images in a digital format. Considering the time separation, the velocity vectors are computed in each interrogation window by determining the averaged particles displacement.

The particle displacement is computed by using the statistical techniques based on cross-correlation as Figure 6 indicates. The maximum value of this cross-correlation shows a strong peak displaced from the origin corresponding to the averaged motion of the particles contained within the interrogation window. This correlation peak was located with sub-pixel accuracy by fitting a Gaussian curve [28].

In the last years, the process to compute the cross-correlation algorithm has been accelerated by means of advanced computational methods [27]. The most important feature of actual implementation of digital PIV (DPIV) technique is the use of 2D fast Fourier transform (FFT) to simplify and speed up the computational process [24]. Taking into account the similarity between spatial cross-correlation and convolution, the cross-correlation of two functions $g_1(x, z)$ and $g_2(x, z)$ is,

$$
\Phi_{\text{cross}}(x, z) = g_1(x, z) \circ g_2(x, z) = g_1(x, z) \otimes g_2(-x, -z)
$$

Figure 5: Interrogation window example.
Where $\phi_{\text{cross}} (x, z)$ is the cross-correlation function, $\circ$ is the cross-correlation operator and $\otimes$ is the convolution operator.

Regarding Fourier transform properties,
\[
\mathcal{F} [g_1(x, z) \otimes g_2(-x, -z)] = G_1(\zeta, \eta) \cdot G_2(-\zeta, -\eta)
\]
(2)
\[
\mathcal{F} [g_2(-x, -z)] = G_2(-\zeta, -\eta) = G_2^*(\zeta, \eta)
\]
(3)

Where $G_2^*$ is the complex conjugate function of $G_2$ and $\zeta, \eta$ are the components in the correlation plane.

Hence, the particle displacement can be obtained using equation 4.
\[
\Phi_{\text{cross}} (x, z) = \mathcal{F}^{-1} [G_1(\zeta, \eta) \cdot G_2^*(\zeta, \eta)]
\]
(4)

Where $\mathcal{F}^{-1}$ is the inverse Fourier transform and $G_1$ and $G_2$ are the Fourier transforms of the intensity distributions of each pair of images as Figure 6 shows.

Figure 6: Cross-correlation process and results file.

Regarding the data file organization and the experimental conditions ($\Delta t$, magnification...), the velocity vector coordinates are indicated for each centre of the interrogation window. This small regions can be related with the grid nodes for CFD. Hence, it is possible to perform a validation process by comparing the results obtained in both cases. From this validation, it is possible to compute the error obtained in CFD codes.
5 RESULTS

As stated, three sections were tested along the spanwise of the ski-jump ramp. The information obtained from PIV experiments allowed to compute the non-dimensional velocity ($\bar{U}$) using the velocity components ($u$, $w$) and the wind tunnel velocity ($U_\infty$) at each case condition, as equation 5 indicates.

\[
\bar{U} = \left[ (u^2 + w^2)^{0.5} \right] / U_\infty
\]  

Turbulence intensity ($TI$) maps were obtained from velocity standard deviation components ($\sigma_u$, $\sigma_w$).

\[
TI = \left[ (\sigma_u^2 + \sigma_w^2)^{0.5} \right] / U_\infty
\]  

Figure 7.a shows the non-dimensional velocity maps at the three sections. In addition, the streamlines were also represented to provide velocity flow structure and recirculation bubble configuration. The turbulence intensity maps are plotted in Figure 7.b.

Figure 7: PIV maps at different sections of the ski-jump ramp (a) non-dimensional velocity (b) turbulence intensity.
Notice that, the lateral effects reduce the low velocity region close to the edges of the ski-jump ramp with respect to the central section (S2). It might be related with the non-symmetry of the ship, which also affects to the flow configuration resulting in a lower recirculation bubble height at the port of the ship. However, the length of the bubble at section 1 is bigger than in section 3. As it can be observed, the incoming flow is slightly affected upstream by the ski-jump ramp presence.

Regarding turbulence intensity (see Figure 7.b), the central section has a bigger turbulent shear layer than the lateral ones. In addition, section 1 presents a remarkable structure difference which could be due to the sharp lateral edge. However, section 3 has an intensity value reduction which can be related to the sponson presence which is characterized by a curved edge that might reduce the lateral effects.

Figure 8 shows the location of reattachment point and the recirculation bubble centre in x-y plane. The reattachment point is defined as the point where the velocity direction over the ski-jump ramp changes and is equal to zero [29]. The recirculation bubble centre is the point where the velocity is zero inside the recirculation bubble region and where streamlines collide. Notice that, y-axis is non-dimensionalized using $b$ parameter which is the spanwise size. The representation of these variables allow to show the lateral effect over the recirculation bubble configuration. Both variables are not directly related although lateral reduction is observed. Despite the high reduction observed in section 3, the reattachment point location for section 1 is lower. The recirculation bubble centre at both sides is located at same horizontal position.

**Figure 8**: Recirculation bubble centre and reattachment point.
6 CONCLUSIONS

The short runway available in aircraft carriers implies the need of additional systems to help aircraft to take-off as the ski-jump ramp that increases the aircraft lift resulting in successful take-off operation. Ahead wind condition is the standard configuration during aircraft performances. The ski-jump ramp presence at this wind condition generates aerodynamic adverse effects for aircraft operations and consequently, pilots have to handle with workload increment and low safety levels during on board operations. Hence, the study of this problem has become more common by using wind tunnel tests and computational methods. Moreover, particle image velocimetry technique has shown to be a powerful tool in the investigation of this kind of flows. In addition, PIV presents useful results for CFD validation.

Experimental tests presented in this papers have been carry out in the facilities of INTA. A closed-circuit wind tunnel with an open test section was used to perform PIV tests. The aircraft carrier model presents a non-symmetric structure which means that aircraft would deal with 3D effects. Three sections along the spanwise were studied to provide 3D information. Furthermore, two PIV captures at each section were developed to provide upstream information of the incident flow.

The results have shown the presence of a 3D flow structure formed by a recirculation bubble located above the ski-jump ramp and bounded by a highly turbulent and non-stationary shear layer. The recirculation bubble has different configuration at each section under study proving three-dimensional nature of the flow over the ramp. Furthermore, the lateral sides are affected by different structures, so the recirculation bubble changes in each section. Turbulence intensity maps also provide turbulent shear layer variations in size. In addition, section 1 presents a region with high turbulence values that might be related with the lateral structure.

PIV maps were analysed to study the centre of the bubble vortex height and the reattachment point of the bubble. Results showed higher values in the central section (S2). Values of these variables in the port section (S1) are lower and they show a slightly asymmetry with respect to the values measured in starboard section (S3).

Finally, we can conclude that the aerodynamic flow in the aircraft carrier ski-jump was experimentally investigated by means of particle image velocimetry (PIV). The results have demonstrated the capability of PIV resolving the 3D flow structure developed over the ski-jump ramp and providing insight and data to be used in the validation process of CFD numerical codes.
REFERENCES


LARGE SCALE CFD MODELING OF WAVE PROPAGATION INTO MEHAMN HARBOR

Weizhi Wang*, Hans Bihs†, Arun Kamath†, and Øivind Asgeir Arntsen†

* Department of Civil and Environmental Engineering
Norwegian University of Science and Technology
Campus Gløshaugen, 7491 Trondheim, Norway
e-mail: weizhi.wang@ntnu.no, web page: www.reef3d.com

† Department of Civil and Environmental Engineering
Norwegian University of Science and Technology
Campus Gløshaugen, 7491 Trondheim, Norway
e-mail: hans.bihs@ntnu.no, web page: www.reef3d.com

Key words: Large Scale, Irregular Bottom, CFD, REEF3D

Abstract. Ocean wave propagations into harbours are large scale phenomena with complex wave transformations. Computational fluid dynamics (CFD) has the advantage of capturing most physics with few assumptions. It has also been successfully applied in marine engineering and coastal engineering. Altogether, CFD is considered to be an ideal tool to analyse the wave propagation at harbours. The most prominent limitation of CFD application is the high requirement of computational resources. However, with increasing computational resources, CFD is becoming an attractive alternative. One of the challenges in large scale CFD simulation is to generate the realistic waves. An extended flat-bottom wave generation zone tends to interrupt the continuity of the sub-sea terrain and introduces unrealistic wave transformations. To locate the wave generation zone far from the topography is one remedy, but it demands more computational resources. Therefore, generating waves over an irregular bottom is of particular interest in CFD simulations. In this paper, the three dimensional large scale numerical simulation of wave propagation into Mehamn harbour is performed with waves generated over an irregular bottom using the open source CFD model REEF3D. The relaxation method is used for the wave generation and absorption. A modified wave generation method considering the local water depths and wave numbers is used in this paper. A study case is performed to demonstrate the effect of the irregular bottom wave generation. Then, the large scale wave propagation into Mehamn harbour is simulated with two different waves generated over the real topography. REEF3D simulates the Mehamn harbour wave propagation by solving Navier-Stokes equations on a staggered grid with the finite difference method. The level-set method is applied to capture the free surface. A fifth-order WENO scheme is applied to the convection terms and a third-order TVD scheme is applied on the transient terms. The topography of the harbour is modelled using the local inverse distance interpolation method. The Mehamn harbour simulations show good wave transformation results and indicate a successful application of wave generation over irregular bottoms.
1 INTRODUCTION

Mehamn harbour is located in Nordkinnhalvya in the Finnmark municipality, Norway. The harbour is connected to the open sea to the north and relatively well protected from waves coming from the east and the west. The geography of the harbour is shown in Fig. 1. Commercial cargo vessels and large fishing vessels dock in the outer part of the harbour and smaller vessels are moored in the inner part of the harbour. In order to have calmer wave conditions for the mooring of the vessels, Finnmark municipality and the Norwegian Coastal Administration decide to arrange two breakwaters along the harbour. Therefore, an accurate modelling for the wave propagation into the harbour is demanded to optimise the effect the breakwaters.

The Norwegian meteorology institute conducted the measurements of the wave data in the offshore region outside Mehamn harbour during 1955 to 2006 [1]. Based on the measurements, SINTEF[1] performed a model test on the wave propagations into the harbour with a scale factor of 1:80. Several numerical wave models have been developed to simulate the coastal waves, such as Simulating Waves Near shore (SWAN)[3] developed by Delft University of Technology. As a third generation wave model, SWAN has been successfully used in predicting significant wave heights and statistical wave properties. However, the phase-averaged results from SWAN are not able to show the details of wave transformations. A phase-resolved model is needed to show the wave propagation patterns and transformation phenomena, such as diffractions and refractions.

Computational Fluid Dynamics (CFD) is able to capture the details and complexities in a fluid domain in an accurate way with less assumptions. Therefore, CFD has been used in a wide range of hydrodynamic applications and has shown great potential for further development. One of the challenges of applying CFD on coastal waves is the high demand of the computational resources. However, with the increasing capacity of the computational resources, CFD can be applied on large scale simulations. REEF3D [4] is a CFD model with specialisation on wave hydrodynamics developed at Norwegian University of Science and Technology (NTNU). The model has been applied to a large range of topics within wave hydrodynamics, such as floating body dynamics in waves[5], breaking waves[6] and breaking wave-structure interactions[7]. The simulations give high-resolution phase-resolved results which present remarkable details of different wave phenomena. In this paper, REEF3D is used to simulate the large scale wave propagation into Mehman harbour.

In order to have phase-resolved results, a good wave generation must be ensured. With the intermediate wave condition in Mehamn harbour, the wave properties are influenced by the water depth variations. Extending the wave tank to arrange a flat bottom wave generation zone might cause unnecessary and unphysical wave transformations because of the sudden geometrical change from the flat bottom to the complex topography. Arranging a flat bottom wave generation zone several wave lengths away from the topography change is a remedy but requires more computational resources. Therefore, a method of generating waves over an irregular bottom with intermediate water condition is needed. With intermediate water condition, the analytical wave velocities and wave lengths are functions of the local wave depth (d) and the wave number (k).
Thus a modified wave generation method with the consideration of \( d \) and \( k \) is used in this paper.

One study case is performed to demonstrate the effect of the irregular bottom generation. Then the 3D large scale CFD simulations of wave propagation into Mehamn harbour are performed with the waves generated over the real topography. The simulations are performed with two different wave conditions and the wave transformation phenomena are compared and analysed.

2 NUMERICAL MODEL

2.1 Governing Equations

Water wave hydrodynamics are well described by the three dimensional incompressible Navier-Stokes equations, as shown in Eqn. (1) and Eqn. (2). For large scale wave modelling, the turbulence terms are neglected from the equations.

\[
\frac{\partial u_i}{\partial x_i} = 0 \quad (1)
\]

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \nu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + g_i \quad (2)
\]

where \( u \) is the velocity, \( \rho \) is the fluid density, \( p \) is the pressure, \( \nu \) is the kinematic viscosity and \( g \) the acceleration due to gravity.

REEF3D solves the Navier-Stokes equations on a staggered grid with the finite difference method and high-order discretisation schemes. The conservative fifth-order weighted essentially non-oscillatory (WENO) [8] scheme is applied on convective terms, resulting in better robustness and smoother flux. The third-order Total-Variation-Diminishing (TVD) Runge-Kutta scheme
[9] is used on time-dependent terms in both Navier-Stokes equations and level-set equations. The Corant-Friedrichs-Lewy (CFL) criterion is applied with adaptive time steps. HYPRE library [11] is used to solve the pressure from the Poisson equation. Furthermore, REEF3D chooses BiCGStab [12] as the iterative solver and the geometric multi-grid PFMG[13] as the pre-conditioner. The combined solver solves the poisson equation very well at large scale simulations.

The free surface is modelled with the level-set method[14]. The method applies a signed distance function denoted as the level-set function $\phi(\vec{x}, t)$ to capture the change of phases at the free surface. As shown in Eqn. (3), the function always equals to zero at the interface of the two phases and shows different signs in each phase. As a result, REEF3D is able to capture complex free surface accurately.

$$
\phi(\vec{x}, t) = \begin{cases} 
> 0 & \text{if } \vec{x} \in \text{phase 1} \\
= 0 & \text{if } \vec{x} \in \Gamma \\
< 0 & \text{if } \vec{x} \in \text{phase 2}
\end{cases}
$$

The bathymetry of the harbour is described by a large set of scattered points. The scattered points are interpolated using the inverse distance weighting method [17] to form the continuous sub-sea topography. The improved local inverse distance weighting method [18] is adopted in REEF3D considering the cost efficiency in terms of time consumption. Instead of a global interpolation regardless of the distance from the points $(x_k, y_k)$ to $(x, y)$, the improved method uses the weight function that only considers the contribution of the points within a certain radii, as shown in Eqn. 4. By assigning the radii as a function of $k$ and using a search technique to find the nearest points near $(x_k, y_k)$, the method is much faster with due accuracy. The irregular bathymetry and coastal line of the harbour are interpolated into a separate level set function. The function is used to distinguish solid from fluid. This approach has the advantage of being very flexible when complex geometries are encountered.

$$W_k(x, y) = \left[\frac{(R_w - d_k)_+}{R_w d_k}\right]^2$$

where $R_w$ is the radii of influence and $d_k$ denotes the Euclidean distance between $(x_k, y_k)$. The relaxation method [15] is applied to generate and absorb the waves. The velocity, pressure and the free surface are ramped up to the analytical values from the wave theories in the wave generation zone before the waves enter the computational domain. In the numerical beach, a reverse process is carried out so that the velocities are reduced to zero, the pressure is damped to the hydrostatic pressure and the free surface is relaxed to the still water level. The wave
The generation and absorption process is described in Eqn. (5). The $\Gamma(\tilde{x})$ term in Eqn. (5) is called the relaxation factor, which is described in Eqn. (6) by Jacobsen[16].

$$\Phi(\tilde{x})_{\text{relaxed}} = \Gamma(\tilde{x})\Phi_{\text{analytical}} + (1 - \Gamma(\tilde{x}))\Phi_{\text{computational}}$$ (5)

where $\Phi$ stands for different parameters in the fluid, including horizontal velocity $u$, vertical velocity $w$, pressure $p$ and level-set function $\phi$.

$$\Gamma(\tilde{x}) = 1 - \frac{e^{(\tilde{x}^2)} - 1}{e - 1} \text{ for } \tilde{x} \in [0; 1]$$ (6)

where $\tilde{x}$ is scaled to the length of the relaxation zone.

With the intermediate water condition, wave properties change with the water depth $d$. As an example, the analytical values of the velocity components of the linear wave theory are given in Eqn. (7). As described in the equations, the velocities are functions of local water depth $d$ and local wave number $k$. Therefore, in the wave generation zone, $d$ and $k$ are not constant numbers but matrices $d(i, j)$ and $k(i, j)$ associated with the $x$ and $y$ coordinates. As a result, the wave properties are ramped up to different analytical values within the wave generation zone.

$$u(x, z, t) = \frac{\pi H \cosh [k (z + d)]}{T \sinh (kd)} \cos \theta$$

$$w(x, z, t) = \frac{\pi H \sinh [k (z + d)]}{T \sinh (kd)} \sin \theta$$ (7)

where $H$ is the wave height, $L$ is the wavelength, $T$ is the wave period, $\omega$ is the angular wave frequency, $d$ is the water depth and $z$ is the vertical coordinate measured from the still water level $z = 0$. $k$ is derived from dispersion relation as a function of $\omega$ and $d$.

## 3 STUDY CASE OF IRREGULAR BOTTOM WAVE GENERATION

A study case is performed to investigate the effect of the modified irregular bottom wave generation method. Two wedges along the wave propagation direction (x-axis) with different initial heights are arranged at the bottom of the wave tank to mimic the depth variation in both $x$ direction and $y$ direction. A two-wave-length numerical beach is located at the end of the tank to absorb the waves. Two wave velocity probes are arranged in the middle of the wave generation zone and in the middle of each wedge in each case. Two wave height gauges are arranged to be 50 m away from the beginning of the bottom topography and at the centre of each wedge in the transverse direction. A linear wave is selected to conduct the study. The chosen wave has a wave height of 1.5 m and a wave period of 15 s. The design of the bottom geometry, the locations of the velocity probes and the wave height gauges and the arrangements of the wave generation zones are all shown in Fig. 2. The velocity profiles at the probes in the wave generation zone and the wave heights at the wave gauges are compared between the irregular bottom wave generation case (IBG) and the extended flat bottom wave generation case (FBG). The mesh convergence study is conducted with the FBG case. The average wave heights at gauge 1 between 220 s and 250 s with different mesh sizes are compared in Fig. 3. The results from all mesh sizes share very similar values with only an error of 0.5 % between
Figure 2: Numerical tank set-up for the study case: (a) the configurations and arrangements of wave velocity probes $P_i$ and wave gauges $G_i$ in the cross section (b) the configurations and the arrangements of wave velocity probes $P_i$ and wave height gauges $G_i$ in the longitudinal direction, $i=1,2$. EF is the numerical beach, AB is the wave generation zone for the IBG case, A’B’ is the wave generation zone for the FBG case. D is the bottom geometry thickness, d is the wave depth.

dx=1 m and dx= 0.8 m cases. Therefore, the mesh size in this study is chosen to be 1 m. The velocity profile at the wave probe P1 and the wave height record at the wave gauge G1 from the IBG case are compared with the counterparts from the FBG case in Fig. 4. The local water depth in the IBG case changes the velocity profile at P1 and as a result the wave height and phase are also slightly different in the computational zone at G1.

4 SIMULATION OF MEHAMN HARBOR

The measurements described in the SINTEF report [1] show that the highest waves come from 345º to 45º. In this paper, it is considered to be the most dangerous situation when the highest waves come directly into the entrance of the harbour, i.e. when the waves come from 345º. Therefore, the original bathymetry is rotated 15º to the east to have the right wave direction. It is stated in the SINTEF report [1] that the 100 year return period extreme wave has $H_{s100}$ of 4.5m and $T_{p100}$ of 15s. NORSOK [2] recommends that the most extreme wave has the wave height of $1.9H_{s100}$. Therefore, in order to simulate the most dangerous wave, the wave height is chosen to be 9 m. With the same wave height, two regular waves are studied with different wave periods. One wave (W1) has the wave period calculated from Eqn. (8) [2], which corresponds to the extreme design wave. Another wave (W2) has the same wave period as $T_{p100}$, which describes the waves contributing the most wave energy. The 5th-order Stokes wave theory is used for W1 while the 2nd-order Stokes wave theory is applied to describe W2.
Figure 3: Mesh convergence study of the irregular bottom wave generation study case

Figure 4: (a) Comparison of velocity profiles at probes P1 and P1' at t=250 s, (b) Comparison of the wave heights at the wav gauges G1 and G1'

The smooth bottom topography obtained from the local inverse distance interpolation method based on a large set of scattered data is shown in Fig. 5. The mesh size is 4m, which gives about 35 and 80 cells per wave length for each wave. The black box in Fig. 5 shows the wave generation zone, which is one wave length long for both wave conditions. There is no numerical beach implemented in the wave thank because the harbour has a closed domain and the region at the end of the tank outside the harbour has shallow water condition, the wave transformation phenomena is better illustrated by including the shallow water part instead of damping the waves out deliberately. The left and right side of the tank have symmetric boundary condition to reduce the reflection from the boundaries.

\[
\sqrt{6.5H_{100}} \leq T \leq \sqrt{11H_{100}}
\]  

(8)

The simulation results for W1 and W2 are shown in Fig. 6 and Fig. 7 respectively. The waves start to refract right after they enter the computational zone because of the deep trench at the beginning of the domain. From both velocity graph and surface elevation graph, one is able to observe that the reflected waves from the peninsulas at the sides of the harbour radiate to the
surroundings. The waves pass through the narrow channel at the entrance of the harbour and curve towards the far end of the harbour. The mean wave level rises up in the shallow area of the harbour and waves dissipate energy resulting in smaller wave amplitude inside the harbour. The various wave transformation phenomena are well visualised and represented in the 3D large scale CFD simulation. The details of the wave patterns facilitate the designs of the breakwaters.

5 CONCLUSIONS

In this paper, the local inverse distance interpolation method is successfully utilised to generate the complicated bottom topography at Mehamn harbour. The irregular bottom wave generation method is also successfully applied to generate regular waves for a large scale domain. The CFD simulations at Mehamn harbour are performed with two different wave conditions and both show high-resolution phase-resolved results with good visualisation. The wave transformations into the harbour are well represented in the simulation results. CFD method is proved to be able to simulate large scale coastal waves and REEF3D is shown to have satisfactory capability of performing such simulations. Further studies of irregular wave CFD simulation at Mehamn harbour and the validation against experimental data will also be performed in the future research.
ACKNOWLEDGMENT

This study has been carried out under the E39 fjord crossing project (No. 304624) and the authors are grateful to the grants provided by the Norwegian Public Roads Administration. This study was supported in part with computational resources at the Norwegian University of Science and Technology (NTNU) provided by NOTUR, http://www.notur.no.

REFERENCES


Figure 7: Free surface velocity (a) and free surface level (b) in Mehamn simulation with H=9 m and T=15 s (W2) at simulation time of 200 s


Weizhi Wang, Hans Bihs, Arun Kamath, and Øivind Asgeir Arnsen


NUMERICAL STUDY OF WAVE TRANSFORMATION USING THE FREE SURFACE RECONSTRUCTION METHOD

ANKIT AGGARWAL*, MAYILVAHANAN ALAGAN CHELLA*, HANS BIHS*, CSABA PÁKZODI†, ØIVIND ASEIR ARNTSEN*

*Department of Civil and Environmental Engineering
Norwegian University of Science and Technology
Web page: http://www.reef3d.com/
†SINTEF Ocean
Trondheim, Norway

Key words: Reconstruction Method, Irregular Waves, CFD, Free Surface Elevation

Abstract. The study of irregular wave field is complex due to its random hydrodynamic characteristics. Many experimental studies have been performed in the past to study irregular waves. However, numerical investigations are less time consuming and expensive as compared to the experimental studies. For a good validation of the numerical model, it is essential to reproduce the laboratory waves numerically. The reconstruction of the numerical irregular free surface elevation is necessary because the paddle signal for the wave-maker in experiments is unknown in most of the cases. It is quite challenging to reconstruct the time history of free surface elevation of irregular waves because of the random wave phases and wave periods. In the present work, a numerical investigation is performed using the open-source computational fluid dynamics (CFD) model REEF3D to test and validate the reconstruction of free surface profiles for irregular wave propagation. Two-dimensional irregular waves are generated by super-positioning of the regular wave components. In the current reconstruction approach, the free surface is reconstructed by representing the irregular free surface elevation as a summation of its Fourier components. First, the free surface reconstruction method is tested for irregular waves in a two-dimensional wave tank with constant water depth. The reconstructed free surface elevations shows a good match with the theoretical wave profiles. Further, the method is used to reconstruct the wave transformation over an impermeable fully submerged bar where the complex phenomena such as shoaling and wave breaking occur. The reconstructed numerical free surface elevations along the wave tank are compared with the experimental free surface elevations. The complex phenomena such as shoaling and breaking are represented with reasonable accuracy in the numerical model.

1 INTRODUCTION

Offshore structures like offshore wind turbines are exposed to irregular breaking wave loads. Offshore engineering applications require a detailed study of the wave hydrodynamics for a safer
design. Thus, the study of irregular breaking waves is important for the associated loads. The complexity involved in the different stages of breaking makes it difficult to study and investigate. Many experimental investigations have been performed to study the transformations of irregular waves in shallow and deep water. Vincet et al. [1] investigated the transformation of monochromatic and directional irregular waves passing over a submerged elliptical mound in a controlled laboratory experiment. Beji et al. [2] investigated the irregular wave propagation over a submerged bar. They examined the wave transformation processes such as wave shoaling, breaking, post-breaking and deshoaling under irregular waves. Ting et al. [3] studied the wave and turbulence characteristics in broad-banded irregular waves over a slope. They also investigated the probability distributions for the wave height and peak velocities for the irregular breaking waves.

Computational fluid dynamics (CFD) can be used as an effective tool to investigate the phenomenon like wave breaking in detail. Many researchers in the past used CFD based two phase flow models for simulating breaking waves [4][5][6][7][8]. The complex dynamics evolve during the breaking process such as the formation of overturning wave crests, enclosed air-packet and the splash-up phenomena. Their results revealed important flow structures in breaking waves. Bredmose et al. [9] investigated breaking wave impacts on monopiles with the focused wave groups using CFD. They compared the numerical and theoretical free surface in time-domain by using the linear reconstruction of waves. However, their numerical model under predicts the irregular wave surface. Also, the phases were not correctly represented by the numerical results. The goal of the present work is to study and validate the reconstruction method for free surface with non-breaking and breaking irregular waves. The investigation is performed is for irregular waves propagating over a submerged bar using REEF3D [10]. The numerical free surface elevations are compared with experimental measurements for different wave gauge locations over the bar. The free surface elevation is reconstructed using the Fast Fourier Transformation (FFT) [11]. The numerical model is able to simulate the wave transformation process including shoaling during the different stages of breaking.

2 NUMERICAL MODEL

The present numerical model is based on the governing equations of fluid dynamics: continuity equation and the Reynolds Averaged Navier-Stokes equations (RANS) with the assumption of an incompressible fluid given as:

\[ \frac{\partial u_i}{\partial x_i} = 0 \]  

\[ \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \nu + \nu_t \right] \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + g_i \]  

where, \( u \) is the velocity averaged over time \( t \), \( \rho \) is the fluid density, \( p \) is the pressure, \( \nu \) is the kinematic viscosity, \( \nu_t \) is the eddy viscosity, \( i \) and \( j \) denote the indices in \( x \) and \( y \) direction, respectively and \( g_i \) is the acceleration due to gravity.

The numerical model uses a fifth-order finite difference Weighted Essentially Non-Oscillatory (WENO) scheme in multi-space dimensions for the spatial discretization [12]. The third order TVD Runge Kutta scheme is used for the time discretization [13]. An adaptive time stepping
scheme is used in the numerical model [14]. The present study uses the $k - \omega$ model [15] along with the Reynolds Averaged Navier Stokes (RANS) equation. The level set method is used to capture the free surface [16]. Detailed information about the numerical model can be obtained in Bihs et al. [10]. REEF3D has been used in the past for a wide range of marine applications, such as wave-structure interaction [17], breaking wave forces [18], floating body dynamics [19] and sediment transport [20]. For the reconstruction of irregular waves, the Fast Fourier Transformation (FFT) algorithm is used. The wave amplitudes ($A_k$), angular frequencies ($\omega_k$) and the phase angles ($\epsilon_k$) are computed for the target irregular wave train at the wave generation using FFT. A time series of free surface elevations can be written as a summation of the Fourier components [11]:

$$\eta(t) = \sum_{k=1}^{N} C_k e^{ik\omega t}$$  \hspace{1cm} (3)

where, $C_k$ denotes the Fourier coefficients.

The computed wave amplitudes ($A_k$), angular frequencies ($\omega_k$) and the phase angles ($\epsilon_k$) are given as an input to the numerical model. The first-order irregular waves are generated by the super-positioning of the linear regular waves components [21][22].

3 VALIDATION WITH THEORY

3.1 Computational setup

The numerical model is tested and validated by comparing the numerically reconstructed wave free surface elevation and the theoretical free surface elevation. The numerical simulations are performed in a two-dimensional (2D) numerical wave tank (NWT) without any structures as shown in Fig. 1. The NWT which is 25 m long and 1 m high with a water depth of 0.5 m is used in the simulations. Three wave gauges are placed along the NWT to study the numerical wave propagation and changes in the wave surface elevation and the numerical results are compared with the theoretical results.

The numerical tests are conducted for three different grid sizes $dx = 0.10$ m, 0.05 m and 0.025 m for the grid refinement study. The irregular waves are generated with a significant wave height $H_s = 0.054$ m and a peak period $T_p = 2.5$ s. Fig. 2 presents a comparison of the numerical results with theory.

Figure 1: Setup of the numerical wave tank (side view)
and theoretical irregular wave free surface elevation for three different grid sizes \( dx = 0.10 \text{ m} \), 0.05 m and 0.025 m at different wave gauge locations. It is observed that for the wave gauge located next to the wavemaker (W1), phase differences are observed between the numerical and theoretical results with \( dx = 0.10 \text{ m} \) (Fig. 2(a)). These phase differences are slightly reduced for the grid size \( dx = 0.05 \text{ m} \) and they are reduced to almost zero for the grid size \( dx = 0.025 \text{ m} \). The numerical wave crests and troughs are in a good agreement with the theoretical results for all grid sizes.

The wave propagation over constant water depth is studied with two more wave gauges along the NWT. The numerical results show some phase differences with the theory for the wave gauge located at \( x = 10 \text{ m} \) (W2) and 20 m (W3) for \( dx = 0.10 \text{ m} \) and 0.05 m. With the grid size \( dx=0.025\text{m} \), the theoretical waves are accurately captured with the correct phases in the numerical simulations (Figs. 2 (c) and 2(e)). Figs. 2(b), (d) and (f) present the irregular wave surface elevation for a longer time series at three different wave gauge locations with the grid size \( dx = 0.025 \text{ m} \). It is noticed that the numerical model is able to accurately reproduce the theoretical irregular wave train (Figs. 2(b)(W1), 2(d)(W2) and (f)(W3). The wave generation, wave propagation and the numerical wave crests and wave troughs are computed with good accuracy. Thus, it can be concluded that the numerical model is able to accurately reconstruct the irregular waves. Fig. 3 presents the simulated free surface changes with velocity magnitude (m/s) variation during the propagation of the irregular wave train. A scaled up view of the wave free surface elevation shows the irregularity in the wave train.

4 IRREGULAR BREAKING WAVES OVER A SUBMERGED BAR

4.1 Setup of the numerical wave tank

In this section, the numerical simulations are performed in a 2D NWT to study the wave propagation and transformation of irregular breaking waves over a submerged bar. The numerical results for the free surface elevation are compared with the experimental results in time-domain [2]. The numerical wave tank is 22 m long and 0.8 m high with a water depth of 0.4 m. A submerged trapezoidal bar with a weather side slope of 1:20 and 2 m horizontal crest followed by a 1:10 lee side slope is placed at a distance of 6 m from the wave generation. The JONSWAP spectrum is used for irregular wave generation in the experiments. The numerical setup is shown in Fig. 4. In the numerical setup, three different wave gauges are placed along the numerical wave tank (NWT) at \( x = 6.0 \text{ m}, 11 \text{ m} \) and 13 m. The numerical tests are performed with input of significant wave height \( H_s = 0.054 \text{ m} \) and peak time period \( T_p = 2.5 \text{ s} \).

4.2 Results

Fig. 5 presents the comparison of the numerical and experimental wave free surface elevation over time at different wave gauge locations. Numerical irregular wave train is in a good agreement with the experimental data measured at the wave gauge located at the toe of weather side slope (W1) (Fig. 5(a)). When the waves propagate over the bar with decreasing water depth, it leads to shoaling. The wave crests and wave troughs show a good match with the experimental results. This complex shoaling process over a submerged bar is well captured in the numerical model as shown in Fig. 5(b) (W2). When the waves propagate over the flat part of the bar, the
Figure 2: Comparison of numerical and experimental wave free surface elevation (m) over time (sec) at a) W1 (shorter time series with different $dx$) b) W1 (longer time series with $dx = 0.025$ m) c) W2 (shorter time series with different $dx$) d) W2 (longer time series with $dx = 0.025$ m) e) W3 (shorter time series with different $dx$) f) W3 (longer time series with $dx = 0.025$ m)
water depth is significantly reduced. The wave crests become sharp and eventually the waves break (Fig. 5(c)). The numerical results show a good comparison with the experimental results for the wave gauge located at $x = 13$ m (W3) over the flat bed. Thus, the numerical results of present study show that the numerical model is able to reconstruct the breaking irregular waves propagating over the submerged bar accurately.

5 CONCLUSIONS

The numerical model REEF3D is used to validate the wave reconstruction method for irregular waves. The waves are reconstructed using the Fast Fourier Transformation (FFT). First, a grid refinement study is conducted to study the effects of grid size on numerical results. The numerical simulations are carried out in a two-dimensional (2D) wave tank without any structures. The grid refinement study evinces that the results with the grid size ($dx = 0.025$ m) show a good agreement with the theory. The wave amplitudes and phases are correctly reproduced in the numerical model. Next, a complex case of irregular breaking waves over a submerged bar is investigated. The free surface elevations computed at three different locations along the bar are compared with the experimental results. A good match between the experimental and numerical results shows that the correct inlet wave is reproduced in the numerical model. The numerical wave amplitudes and wave phases during the wave transformation processes such as shoaling and breaking are represented in the numerical model with reasonable accuracy.
Figure 5: Comparison of numerical and experimental wave free surface elevation (m) over time (sec) at a) W1 b) W2 c) W3

ACKNOWLEDGEMENTS

The research work has been funded by the Research Council of Norway through the project "Hydrodynamic Loads on Offshore Wind Turbine Substructures" (project number: 246810). The authors gratefully acknowledge the computing time granted by NOTUR (project number: NN2620k).

REFERENCES


891


892
ON THE DESIGN OF BLOCK PRECONDITIONERS FOR MARITIME ENGINEERING

C.M. KLAIJ*, X. HE† and C. VUIK‡

* Maritime Research Institute Netherlands
P.O.Box 28, 6700AA Wageningen, The Netherlands
email: c.klaij@marin.nl, web page: http://www.marin.nl/

† Institute of Computing Technology, Chinese Academy of Sciences
No.6 Kexueyuan South Road Zhongguancun Haidian District Beijing, P.R.China 100190
email: hexin2016@ict.ac.cn, web page: http://english.ict.cas.cn/

‡ Delft University of Technology
Mekelweg 4, 2628CD Delft, The Netherlands
email: c.vuik@tudelft.nl, web page: http://www.ewi.tudelft.nl/

Keywords: incompressible flow, Reynolds-averaged Navier-Stokes, segregated and coupled solvers, block preconditioners, Schur complement approximation

Abstract. The iterative error can be an important part of the total numerical error of any Computational Fluid Dynamics simulation when the iterative convergence stagnates or when loose convergence criteria are used. In the quest for better iterative convergence of CFD simulations, we consider the design of iterative methods for the Reynolds-averaged Navier-Stokes equations, discretized by finite-volume methods with cell-centered, co-located variables. The central point is the approximation of the Schur complement (pressure matrix) in the block factorization of the discrete system of mass and momentum equations. We show particular approximations of these blocks that yield either segregated solvers or block preconditioners for fully coupled solvers. The performance of these solvers are then demonstrated by computing the flow over a flat plate and around a tanker on both structured and unstructured grids. We find that iterative convergence to machine precision is attainable despite the high Reynolds numbers and mesh aspect ratio’s. Improved approximations of the Schur complement do result in improved convergence rates, but do not seem to pay-off in terms of total cost compared to the basic SIMPLE-type approximation.

1 INTRODUCTION

Applications of Computational Fluid Dynamics (CFD) in maritime engineering typically involves incompressible, turbulent flow around complex geometries. Most commercial and open-source CFD packages rely on the Reynolds-averaged Navier-Stokes (RaNS) equations to model such flows, on finite-volume methods with co-located variables to discretize these equations, and on SIMPLE-type methods to solve the resulting discrete systems [1, 2, 3]. Various procedures can be applied to quantify the total numerical error of a CFD simulation before making design
choices [4, 5, 6]. The total numerical error consists of a discretization error that can be reduced by grid refinement and of an iterative error that can be reduced by tightening the convergence criteria. Unfortunately, as the grid is refined, the convergence rate of methods such as SIMPLE typically deteriorates to the point where even loose criteria are out of reach. Therefore, in this paper, we focus on alternative iterative methods.

Two alternatives emerge from the vast literature on scalable methods for the incompressible Navier-Stokes equations. Both share the idea of solving the coupled mass-momentum system (instead of solving a SIMPLE-type decomposition), but differ in the way this is achieved. One way is to interlace the velocity and pressure unknowns to form a system of small 4-by-4 blocks. The small blocks on the diagonal are easily inverted, which can be exploited in multigrid methods [7, 8, 9] or in Krylov subspace methods [10]. The other way is to separate the velocity and pressure unknowns to form a system with 4 large blocks and exploit this structure in Krylov subspace methods with block preconditioners. This seems to be a popular choice in the finite-element context (see [11] for an overview), but less so in the finite-volume context, although some results are found in [12] for RANS on co-located grids. On staggered grids, we find results for the Stokes and Oseen equations in [13, 14, 15].

Our research is focused on the design of efficient preconditioners for industrial CFD applications. We have shown in [16, 17] that SIMPLE-type methods can effectively be used as preconditioners, resulting in significant savings of CPU time compared to their standard usage as solvers. In [18], we have shown that the stabilization which is typically used for finite-volume methods with co-located variables to avoid spurious pressure oscillations is particularly favorable for SIMPLE-type methods. Furthermore, we have ported the Augmented Lagrangian Preconditioner [19], which is popular in finite-element context, to the finite-volume context in [20] with promising results for academic test cases.

In this paper, we consider Schur complement preconditioners that are readily available in PETSc [21] once the coupled mass-momentum matrix is provided. Many variations are possible, ranging from the exact Schur complement preconditioner to SIMPLE-type preconditioners and any approximation in-between. We apply three variations to test cases that are relevant for maritime engineering and report iteration counts which, together with a basic cost model, serve as a measure for potential savings in CPU time.

The paper is thus organized in two parts: first a description of the overall algorithm, the mass-momentum matrix, the preconditioners and their relative cost, then followed by examples of the preconditioners’ performance for the three test case.

2 ITERATIVE SOLUTION OF THE RANS EQUATIONS

2.1 Non-linear iteration and coupling

To solve the Reynolds-averaged Navier-Stokes equations we start by linearizing the momentum equations. Since the non-linearity comes from the convective term \( \text{div}(\rho \tilde{v} \tilde{v}) \), it can be linearized by assuming that the mass flux \( \rho \tilde{v} \) is known from a previous iteration. For finite-volume discretization with co-located variables, computing this mass flux involves the pressure-weighted interpolation (PWI) of the velocity, symbolized here by writing \( \tilde{v} \) instead of \( v \). The PWI effectively couples the mass and momentum equations and thereby avoids spurious pressure oscillations. Furthermore, the coupling between the Navier-Stokes equations and the turbulence...
model takes place through the convective terms \( \text{div}(\rho \mathbf{v} \phi) \) in the turbulence transport equations where \( \phi \) is a quantity such as turbulent kinetic energy, and through the diffusive terms \( \text{div}(\mu_{\text{eff}} (\text{grad } \mathbf{v} + (\text{grad } \mathbf{v})^T)) \) in the momentum equations. In these terms, the effective viscosity \( \mu_{\text{eff}} \) is the sum of the (constant) dynamic viscosity \( \mu \) and the (variable) turbulent eddy viscosity \( \mu_t \) provided by the turbulence model. The effective viscosity is assumed to be known when solving the momentum equations. The iterative process of solving the mass and momentum equations with known effective viscosity, computing the mass flux and solving the turbulence equations is repeated until a convergence criterion is met. The solution of the system of mass and momentum equations within this procedure is of the form

\[
x^{k+1} = x^k + x' = x^k + (A^k)^{-1}(b^k - A^k x^k)
\]

with unknowns \( x = [v, p]^T \). Thus, at non-linear iteration \( k \), the value \( x^k \) is corrected by \( x' \), which is the solution of \( A^k x' = r^k \) with \( r^k = b^k - A^k x^k \) the residual of the mass and momentum equations. We are interested in solvers that can exploit the structure of \( A \), which we discuss first.

### 2.2 Structure of the mass-momentum matrix

We consider a finite-volume discretization where the variables are co-located in the cell centers. On a three-dimensional computational mesh with \( N \) cells, we order the \( 4N \) unknowns by equation as \( [(v_1)_{1}, \ldots, (v_1)_{N}, (v_2)_{1}, \ldots, (v_2)_{N}, (v_3)_{1} \ldots (v_3)_{N}, p_1, \ldots, p_N]^T \). The system of mass and momentum equations \( A^k x' = r^k \) then has the following form

\[
\begin{pmatrix}
Q_1 & 0 & 0 & G_1 \\
0 & Q_2 & 0 & G_2 \\
0 & 0 & Q_3 & G_3 \\
D_1 & D_2 & D_3 & C
\end{pmatrix}
\begin{pmatrix}
v_1 \\
v_2 \\
v_3 \\
p
\end{pmatrix}
=
\begin{pmatrix}
r_1 \\
r_2 \\
r_3 \\
r_p
\end{pmatrix}
\]

where each block is a sparse \( N \)-by-\( N \) matrix. Here, \( Q \) contains the contribution from the convection and diffusion, \( G \) from the pressure gradient, \( D \) from the velocity divergence and \( C \) from the PWI.

The block diagonal form of \( Q \) follows not only from the linearization that takes the mass flux \( \rho \mathbf{v} \) from the previous iteration, but also requires the second part of the diffusion term, \( (\text{grad } \mathbf{v})^T \), to be taken from the previous iteration. Notice that the matrices \( Q_i \) are equal because they only contain the convection term \( \text{div}(\rho \mathbf{v} v_i) \) and the remaining diffusion term \( \text{div}(\mu_{\text{eff}} (\text{grad } v_i)) \). To avoid the order barrier [3], higher-order convection schemes are non-linear even though the convection term is linear. Such schemes are implemented in defect-correction form [3, 22] and combined with iteration (1).

The gradient matrix \( G \) and divergence matrix \( D \) are obtained by Gauss’ theorem and linear interpolation. Eccentricity and non-orthogonality corrections [2] are implemented in defect-correction form and combined with (1). The stabilization matrix \( C \) follows from the pressure-weighted interpolation and has the particular form [18]

\[
C = -\frac{1}{dQ} L + \frac{1}{dQ} G
\]
with $L$ the $N$-by-$N$ Laplacian matrix and $dQ$ the diagonal of $Q$. This stabilization is particularly favorable for SIMPLE-type solvers. Notice that $C$, as the difference between two ways of discretizing second-order derivatives, is in some sense close to zero. Solving $A^k x' = r^k$ by the Jacobi iterative method is impossible, because the diagonal elements of $C$ can be zero. For this reason we investigate linear solvers which can be applied even if $C = 0$.

### 2.3 Segregated and coupled solvers

For brevity, we will continue with the following shorthand notation for $Ax' = r$ and consider the LDU factorization of the matrix $A$:

$$
\begin{bmatrix}
Q & G \\
D & C
\end{bmatrix}
\begin{bmatrix}
v' \\
p
\end{bmatrix} =
\begin{bmatrix}
r_v \\
r_p
\end{bmatrix},
\begin{bmatrix}
I & 0 \\
DQ^{-1} & I
\end{bmatrix}
\begin{bmatrix}
Q & 0 \\
0 & S
\end{bmatrix}
\begin{bmatrix}
I & Q^{-1}G \\
0 & I
\end{bmatrix}
$$

with $S = C - DQ^{-1}G$ the Schur complement. The corresponding inverse

$$
\begin{bmatrix}
Q & G \\
D & C
\end{bmatrix}^{-1} =
\begin{bmatrix}
I & -Q^{-1}G \\
0 & I
\end{bmatrix}
\begin{bmatrix}
Q^{-1} & 0 \\
0 & S^{-1}
\end{bmatrix}
\begin{bmatrix}
I & 0 \\
-DQ^{-1} & I
\end{bmatrix}
$$

shows that the solution of $Ax' = r$ involves the solution of momentum sub-systems ($Q^{-1}$) and pressure sub-systems ($S^{-1}$). The main problem is that $S$ is a dense matrix that cannot be explicitly constructed in real-life applications. Therefore, it needs to be approximated, for example by the SIMPLE approximation $\tilde{S} = C - DQ^{-1}G = -\frac{1}{dQ}L$ that exploits the particular form of $C$. If we disregard the upper block, we obtain the following solver $x' = P^{-1}r$ with

$$
\begin{bmatrix}
v' \\
p
\end{bmatrix} =
\begin{bmatrix}
Q^{-1} & 0 \\
0 & \tilde{S}^{-1}
\end{bmatrix}
\begin{bmatrix}
I & 0 \\
-DQ^{-1} & I
\end{bmatrix}
\begin{bmatrix}
r_v \\
r_p
\end{bmatrix} =
\begin{bmatrix}
Q^{-1}r_v \\
\tilde{S}^{-1}(r_p - DQ^{-1}r_v)
\end{bmatrix}
$$

Here, we recognize the first two steps of the SIMPLE method: 1) momentum solve $Q u' = r_v$ and 2) pressure solve $\tilde{S} p' = r_p - Du'$. The third step of SIMPLE is obtained when we also include the upper block in the approximation together with the approximation $Q^{-1} \approx \frac{1}{dQ}$ that was already used for the Schur complement. Other solvers can be devised by making other approximations, the trade-off being that a better approximation is usually more expensive.

Note that $x' = P^{-1}r$ is referred to as a segregated solver: it does not involve the matrix $A$. The matching coupled solver $x' = A^{-1}r$ is obtained by using $P^{-1}$ as a preconditioner, i.e. by solving $P^{-1}Ax' = P^{-1}r$ with a Krylov subspace method such as GMRES. Since the coupled solver computes the corrections $x'$ with the original matrix $A$ instead of the cheaper approximation $P$, we expect faster convergence of the sequence $\{x^k\}$ but also a higher cost per iteration.

Once the matrix $A$ is provided, PETSc can readily construct preconditioners based on the inverse of the LDU factorization with any desired approximation. Here, we will consider three variations$^1$:

- ‘exact Schur’: full factorization with momentum and pressure sub-systems solved to a relative tolerance of $10^{-12}$. Preconditioner for $S$ based on the SIMPLE approximation $\tilde{S}$.

---

$^1$Be careful, similar names in literature can have slightly different meaning.
• ‘approx Schur’: same as exact Schur but with a relative tolerance of $10^{-2}$ instead of $10^{-12}$.

• ‘SIMPLE’: same as approx Schur but with $\tilde{S}$ instead of $S$.

For the mass-momentum system we use FGMRES with any of these preconditioners, for the sub-systems we use GMRES with ILU(0) preconditioning. The exact Schur preconditioner should lead to convergence in two steps [23], which is the main motivation for considering cheaper approximations that are feasible in real-life applications.

2.4 Solver cost model

Wall-clock time may be the obvious measure to discriminate between different solvers but since it highly depends on machine and implementation, we get more insight from a basic cost model.

First consider the cost of the segregated SIMPLE solver. This forms the baseline for evaluating the cost of other solvers. At each non-linear iteration, SIMPLE solves the three momentum equations (‘mom-u’) and the pressure equation (‘mass-p’) to a certain relative tolerance, which we assume to be fixed. Thus, the total cost expressed in solves per non-linear iteration is

- segregated SIMPLE: $3 \text{mom-u} + 1 \text{mass-p}$.

Now consider the cost of using SIMPLE as a preconditioner for a Krylov subspace method that solves the coupled mass-momentum system to a certain relative tolerance in $n_x$ iterations. The preconditioner is applied at each Krylov iteration. Therefore, if all other costs are assumed negligible compared to the cost of the preconditioner, we arrive at a total cost of

- coupled, SIMPLE preconditioner: $n_x \times (3 \text{mom-u} + 1 \text{mass-p})$

per non-linear iteration. Clearly, this higher cost per non-linear iteration only pays-off if the number of non-linear iterations required to achieve convergence is significantly reduced compared to the baseline solver.

Finally consider the cost of the Schur preconditioners. Again these preconditioners are applied at each Krylov iteration of a coupled mass-momentum solver. However, the matrix in the pressure equation is now different. Instead of solving $\tilde{S}p = r_p$ with $\tilde{S} = C - D \frac{1}{dQ} G$ we are now solving with $S = C - DQ^{-1}G$. To quantify the cost of this pressure solve we have to consider it in more detail. In both cases, we solve the pressure sub-system with a preconditioned Krylov subspace method up to a certain relative tolerance in $n_p$ iterations. In both cases, we use the same preconditioner, ILU(0) based on the matrix $\tilde{S}$. Thus, the application of the preconditioner has the same cost. But preconditioned Krylov subspace methods further require the application of the matrix itself and this is where both approaches differ. In the first case, the matrix is readily available. In the second case, applying the matrix involves four steps: multiplication with $G$, three mom-u solves, multiplication by $D$ and subtraction from $C$. If we neglect the multiplications and subtraction, we arrive at a total cost of

- coupled, Schur preconditioner: $n_x \times (3 \times (n_p + 1) \text{mom-u} + 1 \text{mass-p})$
Figure 1: Impression of the grids. Flat-plate grid with 160×120 cells and max aspect ratio of order 1 : 10^4 (left). Tanker grids, unstructured with 176K cells and max aspect ratio of order 1 : 10^2 (middle) and structured with 256K cells and max aspect ratio of order 1 : 10^4 (right).

per non-linear iteration. The $n_x \times n_y$ factor reflects the nesting of a momentum solve within the pressure solve. Clearly, this higher cost only pays-off if the number of linear iterations is significantly reduced compared to the SIMPLE preconditioner.

Thus, by quantifying the number of non-linear iterations and the number of Krylov iterations $n_x$ and $n_y$ in the following numerical experiments, we can already get an impression of the relative cost of the proposed solvers.

3 NUMERICAL EXPERIMENTS

The following numerical experiments were done with a pilot implementation in MARIN’s CFD software package ReFRESCO [24].

3.1 Description of test cases

Flow over finite flat plate The friction resistance coefficient of an infinitely thin plate, expressed as function of the Reynolds number, is known as the friction line and plays an important role in the extrapolation of model-scale results to full scale. The flow over a finite flat plate therefore became a standard test case in maritime engineering. Numerical friction lines for various turbulence models were already proposed in [25] and compared to experimental lines such as the ITTC’57 line. Detailed results for ReFRESCO are found in [26], including a numerical uncertainty analysis with the procedure from [6]. Here, we will reconsider the fully turbulent flow at $Re = 10^7$ on the same series of grids. These block-structured grids are refined near the leading and trailing edge of the plate and spread out in the wake of the plate, see Figure 1. Near the middle of the plate, the cells have an aspect ratio of order $1 : 10^4$.

Flow around model tanker Another important test case in maritime engineering is the flow around the Kriso Very Large Crude Carrier 2, see [27] for a summary of the results from literature and for a detailed study of various turbulence models with ReFRESCO. Results for different segregated and coupled solvers can be found in [17]. Here, we reconsider the model-scale case at Reynolds number $4.6 \cdot 10^6$ on two types of grids, structured and unstructured. The structured grid has 256K cells and the unstructured grid 176K cells, which is very coarse for this
application. The main difference is that the unstructured grid allows local refinement towards the hull using hanging nodes, see Figure 1. The structured grid has a maximum aspect ratio around $1 : 10^4$ and the boundary layer is fully resolved ($\max y^+ \approx 1$) while the unstructured grid has a maximum aspect ratio around $1 : 10^2$ and part of the boundary layer is modeled with wall functions ($\max y^+ \approx 350$).

3.2 Non-linear iteration

Figure 2 illustrates the benefit of the coupled solver: it requires less non-linear iterations to achieve a certain convergence criterion than the segregated solver. The same relaxation parameters were used for both solvers, so the difference in convergence is solely due to the coupled solution of the mass-momentum system. As shown in [17], this difference increases with grid refinement, leading to significant savings in CPU time on finer grids. The coupled solver also proved to be less sensitive to relaxation parameters. Note that the non-linear iterations are more expensive than those of a segregated solver, so the actual saving highly depends on the efficiency of the linear solver and preconditioner for the mass-momentum system, which is the main topic of this paper.

3.3 Linear iteration

Figure 3 shows an example of the convergence of FGMRES with the three different preconditioners discussed in Section 2.3. We see that FGMRES converges to machine accuracy in $n_x = 2$ iterations when the exact Schur preconditioner is used, as predicted by theory. When the approximate Schur preconditioner is used, $n_x = 6$ iterations are needed to reach machine accuracy for the flat plate, $n_x = 12$ for the tanker on the unstructured grid and $n_x = 13$ on the structured grid. The SIMPLE preconditioner does not converge very well in comparison, it only achieves one or two orders of convergence in $n_x = 20$ iterations. Clearly, in terms of linear iterations for the mass-momentum system, there is a lot to be gained by using better approximations than SIMPLE.

However, according to the cost model from Section 2.4 we should also consider the linear iterations of the pressure sub-system involved. For each iteration, the exact Schur preconditioner
Figure 3: Linear convergence of the coupled mass-momentum system at the 10th non-linear iteration with three preconditioners (left) and corresponding number of iterations needed to solve the pressure sub-system (right). Flat plate on 80×60 grid (top), tanker on unstructured grid (middle) and on structured grid (bottom).
(which solves sub-systems to relative tolerance $10^{-12}$) needs more than $n_p = 300$ iterations to solve the pressure sub-system for the flat plate, more than $n_p = 500$ for the tanker on the unstructured grid and more than $n_p = 1700$ for the structured grid. The approximate Schur preconditioner (which solves sub-systems to relative tolerance $10^{-2}$) needs about $n_p = 50$ for the flat plate, $n_p = 80$ for the tanker on the unstructured grid and $n_p = 200$ for the structured grid. The SIMPLE preconditioner (which solves sub-systems to relative tolerance $10^{-2}$ and approximates the Schur complement) needs less than $n_p = 25$ for all cases. Clearly, the reduced number of iterations for the mass-momentum system obtained with the approximate Schur preconditioner comes with an increased number of iterations for the sub-systems.

So far, we considered the solution of the mass-momentum system arising at the tenth non-linear iteration. In the context of a non-linear solver, it is often not necessary to solve this system very accurately and a relative tolerance of 0.01 is already sufficient [17]. Therefore, we continue with this tolerance and considering the first one-hundred non-linear iterations. Figure 4 shows the number $n_x$ of linear iterations needed to solve the mass-momentum system at each non-linear iteration and the average number of iterations $n_p$ needed to solve the pressure sub-system.

Considering that the test cases vary from 2D to 3D, from structured to unstructured and from moderate to high aspect ratio’s, we find the number $n_x$ to be remarkably constant: around 5 for the approximate Schur preconditioner and around 20 for the SIMPLE preconditioner. The number $n_p$ is less constant and depends both on the case and on the non-linear iteration. Extreme differences are found for the tanker on the structured grid: in the first forty non-linear iterations $n_p$ is around 140 for the approximate Schur preconditioner and around 25 for the SIMPLE preconditioner, then these numbers drop to around 60 and 10, respectively. This makes it hard to draw any definite conclusions. Nevertheless, if we assume typical values for the approximate Schur preconditioner to be $n_x = 5$ and $n_p = 50$ we get a total cost per non-linear iteration of

- coupled, approx Schur preconditioner: 765 mom-u + 5 mass-p

while the SIMPLE preconditioner with typical value of $n_x = 20$ gives a total cost of

- coupled, SIMPLE preconditioner: 60 mom-u + 20 mass-p

From these numbers, we tentatively conclude that the reduction of iterations $n_x$ achieved with the approximate Schur preconditioner is not nearly enough to compensate for the momentum solve needed at every iteration of the pressure solve.

One could also conclude that the SIMPLE approximation $\tilde{S}$ is a poor preconditioner for the true Schur complement $S$: a better preconditioner might reduce $n_p$ and shift the cost balance. Furthermore, note that all cases presented here were done on a single process. In parallel, the preconditioner is built from $\hat{S}$ using ILU(0) per sub-domain and Jacobi between sub-domains, which leads to a dramatic increase of $n_p$. A possible explanation is that $\hat{S}$ is sparse, whereas $S$, if it were constructed, would be dense. With parallelization using ILU(0) per sub-domain and Jacobi between sub-domains, the preconditioner built from $\hat{S}$ becomes even more sparse, which could explain the significant loss in performance when used for solving dense $S$. This drop in performance has prevented us from presenting thorough scalability studies.
Figure 4: Linear iterations needed to solve the mass-momentum system with relative tolerance 0.01 (left) and average number of iterations needed to solve the pressure sub-system (right). Flat plate on 80×60 grid (top) and tanker on unstructured grid (middle) and on structured grid (bottom).
4 DISCUSSION

We have compared the behavior of three preconditioners for the solution of the mass-momentum system that arises from the finite-volume discretization with co-located variables of the steady, incompressible RaNS equations, using three test cases that are representative of maritime engineering.

These preconditioners are readily available in PETSc and differ only in the approximation of the Schur complement and in the relative tolerance of the sub-system solves. The main idea was to keep the true Schur complement in the pressure sub-system and use the SIMPLE-type approximation to build its preconditioner. This gives much better performance than replacing the true Schur complement altogether, at least when expressed in linear iterations for the mass-momentum system.

However, the total cost in terms of linear iterations for the sub-systems turns out to be an order higher for the three considered cases. The main reason for this difference is that each pressure sub-system iteration with the true Schur complement involves the solution of the momentum sub-system. For our test cases, this easily leads to hundreds of momentum solves within each pressure solve. Besides, the SIMPLE-type approximation turns out not to be a very good preconditioner for the true Schur complement, especially in combination with block Jacobi parallelization. This has prevented us from presenting grid studies and scalability results in this paper. Future research should therefore aim at understanding the exact causes of these problems.

ACKNOWLEDGEMENT

This research is partly funded by the Dutch Ministry of Economic Affairs.

REFERENCES


ON THE USE OF EULER AND CRANK-NICOLSON TIME-STEPPING SCHEMES FOR SEAKEEPING SIMULATIONS IN OPENFOAM®

Sopheak Seng*, Charles Monroy, Šime Malenica

Bureau Veritas, Marine & Offshore Division
Research Department, 92571 Neuilly-sur-Seine, France
*e-mail: sopheak.seng@bureauveritas.com

Key words: Seakeeping, OpenFOAM, VOF, Crank-Nicolson, Euler

Abstract. The open-source CFD software package OpenFOAM has reached a maturity level such that it is possible to perform seakeeping simulations using the included VOF-based free-surface URANS (Unsteady-Reynolds-Averaged Navier-Stokes) solver (a.k.a. interDyMFoam). This paper describes results of seakeeping tests and the experiences obtained while selecting an appropriate combination of spatial and temporal schemes for wave and seakeeping simulations in a regular head sea condition. Particular attention has been paid to the accuracy level and the convergence rate of temporal schemes Euler and Crank-Nicolson since an accurate temporal discretization is known to be very important for wave propagations. Here the numerical results confirm the need for at least a 2nd-order temporal scheme. To improve the stability and the robustness of the existing Crank-Nicolson scheme we customized the code and performed a numerical experiment on the modified Crank-Nicolson scheme where the off-centering parameter $c_o$ is non-uniformly distributed in the domain. The results show that when using a simple distribution of the $c_o$ parameter the stability of the Crank-Nicolson scheme can be restored without having to degrade significantly the numerical order of the scheme. This new approach allows stable simulations to be performed where the incident wave field is propagated more acceptably with a very small decay and, at the same time, keeps the time step large enough to allow the simulations to finish at a reasonable CPU time.

1 INTRODUCTION

The study of ship motion in waves is commonly performed using tools based on potential flow theory which are very efficient in linear waves but cannot account sufficiently for highly nonlinear waves, strong body nonlinearity, high Froude-number and the viscous effects. In the hope of overcoming these issues simulations based on a free-surface URANS (Unsteady Reynolds-Averaged Navier-Stokes) solver is becoming increasingly more popular since the solver and the necessary seakeeping capabilities are available in commercial codes as well as in open-source codes. One of the critical components for seakeeping simulations is the capability to generate waves and propagate them accurately toward the ship. Hence, one of the preliminary
steps to evaluate a free surface URANS code for its seakeeping capability is to establish practical experiences on how wave systems and perhaps more fundamentally the dynamics and kinematics of the free surface can be simulated efficiently and accurately. Physically, besides the incident wave field there are several important wave systems generated by the presence of the ship and its motion. For example, at a certain combination of low Froude numbers and wave frequencies the radiated and diffracted waves are propagating upstream ahead of the ship creating a complicated interfering pattern with the incident wave field. These wave systems travel very far from the ship. Since most of the free surface URANS codes need a finite computational domain the wave systems may cause stability problems, numerical artifacts at wave generation boundaries and a certain level of artificial wave reflection from the truncated boundaries.

The work done in this paper uses the open-source software package OpenFOAM [1, 2] with a necessary customized third-party wave library based on [3] to introduce waves into the computational domain. The objective of the work is to evaluate from a practical point of view the robustness, the efficiency and the accuracy of the public domain OpenFOAM code for seakeeping simulations. A seakeeping case in a head sea condition in a long crested regular wave is selected for the purpose. The configuration of the seakeeping mesh is selected appropriately to include sufficient refinements within the free surface zone, the wave propagation zone, the wave damping zone and the near-hull local regions. The errors related to the propagation of the regular incident wave field on this selected mesh configuration are first evaluated before performing the full seakeeping simulation. The flow solver uses the FVM (Finite Volume Method) discretization for solving URANS equations and the free surface is captured by a VOF (Volume of Fluid) method. The capability to generate and absorb waves is added according to [3]. The mathematical formulation and the important aspects of this solver are presented in Sec. 2 below. Two test conditions are presented. The first test condition is for the propagations of the incident wave field on a 2D mesh with a similar configuration as for the 3D seakeeping mesh. This test condition shall provide a good insight on how well the simulated incident wave field can be propagated from the wave inlet boundary toward the ship. The second test condition is the 3D seakeeping test performed with similar numerical schemes and configuration taken from the best known 2D wave tests. The general numerical setup is described in Sec. 3 and the results of the simulations are discussed in Sec. 4.

2 NUMERICAL MODEL

The OpenFOAM software package is a collection of C++ libraries which provide core functionalities for solving partial differential equations. One of its most developed functionalities is the FVM discretization using unstructured polyhedral meshes. Several basic solvers are included in the package. The flow solver used in this work is a VOF based incompressible two-phase solver. It solves the URANS equations and a modified transport equation of the phase fraction field. Details related to the basic principle of VOF can be found in [6]. Air and water are considered immiscible and treated as viscous and incompressible. While the computational domain includes both air and water with a density ratio up to approximately 1000, the mass and momentum conservation are formulated under the assumption that the density $\rho$ and viscosity $\mu$ of the effective fluid can be computed using the volume phase fraction field as weighting factor as in
Eq. (1) below:
\begin{align}
\rho &= \alpha \rho_a + (1 - \alpha) \rho_w \\
\mu &= \alpha \mu_a + (1 - \alpha) \mu_w
\end{align}
(1)

where the subindices "a, w" indicate air and water, respectively. The volume phase fraction field \( \alpha \) must be bounded between 0 and 1 since it describes the volume fraction of water in the control volume. Here the surface tension is considered negligible. The governing equations for the laminar model (the turbulence model is runtime selectable) including the modified transport equation read
\begin{align}
\nabla \cdot u &= 0 \\
\frac{\partial \alpha}{\partial t} + \nabla \cdot \alpha u + \nabla \cdot [\alpha(1 - \alpha)u_r] &= 0 \\
\frac{\partial \rho u}{\partial t} + \nabla \cdot (\rho uu^T) - \nabla \cdot [\mu(\nabla u + \nabla u^T)] &= -\nabla p_{rgh} - (g \cdot x) \nabla \rho
\end{align}
(2)
(3)
(4)

The pressure field \( p_{rgh} \) is defined in relation to the static pressure \( p \) as \( p_{rgh} \equiv p - \rho g \cdot x \), where \( g \) is the vector representing the gravitational acceleration and \( x \) is the position vector. The velocity/pressure coupling is resolved in time using a modified PISO algorithm known to the OpenFOAM community as the PIMPLE algorithm which is a unified implementation of the PISO algorithm (pressure-implicit with split-operator [7]) and the SIMPLE algorithm (Semi-Implicit Method for Pressure Linked Equations). Through user-selected run-time controlling parameters the PIMPLE algorithm can be configured to operate in pure PISO or SIMPLE mode. More importantly it enables a combined use of SIMPLE/PISO mode (a.k.a PIMPLE mode) to achieve a more efficient time stepping since in this mode the time step size shall be less constrained than in the PISO mode. The PIMPLE mode has been used in all simulations performed this work.

To keep the \( \alpha \)-field bounded between 0 and 1, Eq. (3) is solved using the semi-implicit MULES solver (Multidimensional Universal Limiter with Explicit Solution). Although MULES has been introduced into the OpenFOAM library almost a decade ago the literature on this solver is sparse. The work done in [8, 9] are examples of recent efforts to describe the MULES solver. The ability of MULES in keeping the \( \alpha \)-field bounded is documented in [9]. The artificial term in Eq. (3) (the third time on the left-hand-side) is added in an attempt to prevent smearing of the interface due to numerical diffusion in the \( \alpha \)-field. This term is designed to compress the interface by adding artificial flux to the equation and requires an empirical model for \( u_r \) (see also [10]), the relative velocity between the air and water, which in theory should be zero but in practice is different from zero due to the numerical smearing and the difficulty of representing the interface exactly in the VOF formulation. The treatment of the transport equation for the \( \alpha \)-field using this artificial term in a combination with the MULES solver can be classified as an algebraic VOF algorithm. While it is possible to obtain good results with this algorithm the conclusion put forward by [9] is that MULES is inferior to geometric VOF algorithms. Since the OpenFOAM software package does not provide other options, the current work uses exclusively the semi-implicit MULES in all simulations.

The FVM discretization of the governing equations is managed by the OpenFOAM core library which follows the standard FVM practice of converting volume integrals to surface integrals using Gauss’s theorem. The integration and interpolation scheme for each term in the
equations are made runtime selectable. These include the time derivative term for the momentum equation and the transport equation, the divergence and the gradient operators. The 2nd-order upwind scheme has been selected for the convective term in the momentum equations and the 2nd-order central differencing scheme for the diffusive term. The \( \alpha \)-field always has a steep gradient near the free surface. Hence, the 2nd-order high-resolution shock capturing scheme \textit{van Leer} is selected for the convective term in the transport equation for the \( \alpha \)-field.

For the time derivative terms OpenFOAM core library provides three different schemes named: Euler, backward, and CrankNicolson. The Euler scheme in OpenFOAM is an implementation of the backward Euler integration algorithm which is implicit and known to be very stable but only 1st-order accurate. The numerical results of wave propagations which are presented and discussed in Sec. 4.1 show that the Euler scheme, although very stable, is 1st-order accurate and is not sufficient for propagating waves from the wave generation boundaries toward the ship. To maintain an acceptable wave condition within the vicinity of the ship the order of the temporal scheme needs to be at least 2nd-order accurate which can be achieved using either backward or CrankNicolson. In OpenFOAM backward is the 2nd-order three-time-steps backward differentiation formula and CrankNicolson is a modified Crank-Nicolson scheme where an off-center parameter with a value between 0 and 1 has been introduced to control the weight toward either standard Crank-Nicolson scheme or the implicit Euler scheme. It appears that the backward or the CrankNicolson schemes can be readily applied to obtain a 2nd-order integration in time. However, the backward scheme is known to be unbounded. Hence it cannot be applied to the transport equation for the \( \alpha \)-field since it will cause extreme difficulty to control a bounded solution of the \( \alpha \)-field to within 0 and 1. The option to use the backward scheme in MULES is simply disabled. The CrankNicolson scheme, on the other hand, can be made bounded and is supported by MULES to obtain a consistent 2nd-order temporal accuracy not only for the momentum equations but also for the transport equation of the \( \alpha \)-field. The simulation performed in this work, however, quickly reveals that the CrankNicolson scheme may cause the numerical solution to have an oscillatory convergence. When the scheme is used directly in FSI (fluid structure interaction) simulations where the body motion is resolved through a partitioning scheme where the flow solver and the body-motion solver are executed alternately (with fixed under-relaxation of the body acceleration) the oscillatory flow solutions are having a strong negative influence on the convergence behavior of the FSI iterations. When the FSI converges, the number of FSI iterations increases significantly. Under certain circumstances the FSI iterations diverges. The oscillatory convergence behavior of the CrankNicolson scheme is well known and the oscillatory behavior is associated with a too large Courant number (\( C \gg 1 \)). Under normal circumstances the Courant number is computed as the global maximum which is conservative since the Courant number is critically high at a few local cells only. These cells are typically inside local refinement regions where the cells are very small due to either physical constraints such as boundary layer flow and the selected turbulence model or technical issues related to automatic mesh generations. The latter seems evident when using automatic mesh generation tools which are based on unstructured split-hexahedra. These are e.g. the freely available \texttt{snappyHexMesh}\footnote{A utility provided by the OpenFOAM software package to generate unstructured 3-dimensional meshes containing mostly hexahedra and spitted hexahedra} or similar commercial alternatives such as HEXPRESS™ or STAR-CCM+. 
Assuming that these critical small cells region are an unavoidable product of the automatic mesh generation tools and the physics of the flow within these regions do not require the cells to be so small it is natural to think of a solution to trade on the unnecessarily high spatial resolution for an increasing numerical stability. One of the easiest solution which is put under a numerical experiment in this paper is to adjust locally the off-center parameter found in the Crank-Nicolson scheme. In OpenFOAM the implementation of this solution is straightforward costing no more than a few lines of codes added to the source code of the flow solver. Here this solution shall be described in detail.

Recall that the standard Crank-Nicolson scheme can be viewed as a half-and-half weighting between the forward and backward Euler-schemes, the weight coefficient can be considered as an adjustable parameter which controls the numerical behavior of the scheme. In OpenFOAM this weighting coefficient has been implemented as described below. Consider the following generic 1st-order ODE (ordinary differential equation)

\[ \dot{y} = f(t, y) \]  

According to the forward Euler scheme the integration from time step \( n \) to \( n+1 \) is approximated by \( y_{n+1} - y_n \approx hf_n \) where \( h = t_{n+1} - t_n \) is the time step size and \( f_n = f(t_n, y_n) \). For the backward Euler scheme the approximation is \( y_{n+1} - y_n \approx hf_{n+1} \). A weighting coefficient \( \gamma = [0, 1] \) has been introduced as follows:

\[ y_{n+1} - y_n \approx \gamma hf_{n+1} + (1 - \gamma)hf_n \]  

For \( \gamma = 1/2 \) the scheme is equivalent to the standard Crank-Nicolson scheme. The scheme can be made equivalent to the forward explicit Euler scheme by selecting \( \gamma = 0 \). For stability reason, however, it is advisable to keep \( \gamma \) between 1/2 and 1, where \( \gamma = 1 \) is equivalent to the fully implicit 1st-order backward Euler scheme. In OpenFOAM \( \gamma \) can be adjusted within the range of \( \gamma = [1/2, 1] \). The adjustment is made available during run-time through an off-center coefficient \( c_o \) which is related to \( \gamma \) as follows

\[ c_o = \frac{1 - \gamma}{\gamma} \quad \text{for} \quad \gamma = [0.5, 1] \]  

From this definition \( c_o \) will have a value between 0 and 1. The standard 2nd-order Crank-Nicolson scheme \( (\gamma = 0.5) \) is specified as \( c_o = 1 \), and the implicit 1st-order backward Euler \( (\gamma = 1) \) as \( c_o = 0 \). It is noted that a 2nd-order accuracy can be expected only by using \( c_o = 1 \) (i.e. \( \gamma = 0.5 \)). Clearly if the simulation runs stable at \( c_o = 1 \) it will not be wise to lower this coefficient. Changing the value of \( c_o \) slightly, says \( c_o = 0.9 \), will increase numerical dissipation and reduce the order toward the fully 1st-order backward Euler. The lost of accuracy shall be judged against the benefit of having an additional numerical dissipation which means better numerical stability. In cases the simulations cannot proceed at \( c_o = 1 \) due to stability reasons this additional dissipation may become very valuable if it restores fully the numerical stability with a minimal and local impact on the flow features of interest. In seakeeping the flow features of interest, preliminarily, are the wave field at the position of the ship. From this point of view it is justifiable to allow a non-constant spatial dependency of the \( c_o \) parameter if the benefit of added numerical stability outweighs the lost of accuracy. Obviously it is not trivial to determine
an optimum spatial dependency for $c_o$. The first aims, however, is not to have a fully optimal distribution for $c_o$ but to establish some evidences showing that the benefit of having a non-constant spatial distribution of $c_o$ can be achieved in practice. The numerical experiments constructed in this paper shall provide more insight on this subject.

3 NUMERICAL SETUP

It is generally acknowledged that seakeeping simulations based on a FVM free-surface URANS solver are computationally very expensive which provides a strong motivation for keeping the cell count low. The seakeeping mesh is generated using unstructured body-fitted split-hexahedra mesh. Waves are introduced into the domain by imposing wave kinematics and free surface conditions at the wave generation boundaries. To prevent wave reflection from the outlet boundary we use an explicit relaxation zone technique based on the description in [3]. A study on this wave generation and absorption technique can be found in [4].

Two test conditions are presented. The first test consists of wave propagations in a two-dimensional wave flume and the second test is for three-dimensional simulations of a ship moving in head waves. A mesh for the 2D case is constructed without the ship to have the same refinement configurations as in the 3D seakeeping case. This 2D mesh essentially is a slice of the 3D mesh through a plane parallel to the symmetry plane of the ship. Therefore the numerical results of the long-crested regular waves on this 2D mesh are considered representative for the results on the 3D mesh. Figure 1 shows an illustration of the 2D mesh used in the wave propagation tests. The mesh resolution shown in this figure has been reduced considerably for visualization purposes. Here it is shown that three isotropic volume refinement regions (Refinement box 1, 2 and 3, see Fig. 1) are defined to obtain a reasonable coarsening toward the outlet boundary and a good mesh resolution within the wave propagation zone and around the ship. Cells are stretched vertically away from the free surface region. The total length of the domain in the wave propagation direction is $4.5\lambda$ where $\lambda$ (see Table 1) is the wave length predicted by the nonlinear stream function wave theory [5]. A region next to the outlet boundary

\textbf{Figure 1:} A sketch of the mesh configuration and the boundary conditions (BC) defined in the 2D wave tests (without the ship). The longitudinal position of the ship is indicated by AP (aft perpendicular), MS (midship) and FP (forward perpendicular). The wave length is $\lambda \approx 11.84$ m c.f. Table 1.
equivalent to $2\lambda$ is reserved for wave damping. According to results published in [3] and [4] the selected length of $2\lambda$ is more than sufficient to prevent wave reflection from the outlet boundary. The resulting 2D mesh has approximately 28000 cells in total and the wave propagation zone has about 190 cells per $\lambda$ and 12 cells per wave height. The width to height cell aspect ratio in this zone is approximately 4.

For the seakeeping simulation the container ship KCS in model scale is selected. The experimental data for a head sea condition is publicly available e.g. in [11] and in the proceeding of the workshop on CFD in ship hydrodynamics in Tokyo Dec. 2nd-4th 2015 [12]. The selected test case is label KCS-CASE-2.10-c5 in [12]. A short summary of the test condition is given in Table 1. The template for the 3D mesh is the same as for the 2D mesh (Fig. 1) including the isotropic volume refinement regions and the stretching toward the free surface zone. Here, the ship is cut out and the cut-cells are refined and fitted to the ship hull. A close-up view of the cells in the vicinity of the hull is shown in Fig. 2. Cells next to the hull within a normal distance of about 4 times the cell length in the wave propagation zone are refined gradually 2 times as shown on the magnified area in Fig. 2. This 3D mesh has been produced using open-source meshing tools including blockMesh, refineMesh, snappyHexMesh and cfMesh which (apart from the latter) are readily available in the OpenFOAM package. These tools are known to have difficulty to produce good quality prism layers on the hull. Hence the simulations are performed without the prism layers and the turbulence model has been de-activated assuming a laminar viscous flow passed the ship. The consequence of this configuration is that the viscous shear stress is not correctly captured by the flow solver which affects strongly the predicted total hull resistance. However, heave and pitch motions are dominated by the vertical forces. The viscous shear stress which contributes mainly to the horizontal forces in the direction of the forward speed therefore will have a negligible effect on heave and pitch motion. The symmetric flow in head sea conditions are exploited such that only half of the ship (and the fluid domain) is modeled. The length, width and height of the 3D computational domain are respectively $\approx (4.5, 1.5, 0.7)\lambda$. The bottom is flat at a water depth of 5.5 m. There are $3\lambda$ from the outlet boundary to the
ship whereas \(2\lambda\) are reserved for wave damping. The wave propagation zone in front of the ship is \(1\lambda\) and it spans from the wave inlet boundary to the ship. From the ship to the side wall of the computational domain there is \(1.5\lambda\). This configuration produces approximately 1.8 millions cells in total.

4 NUMERICAL RESULTS AND DISCUSSION

All simulations performed in this work use a modern HPC (high performance computing) cluster located at Ecole Centrale de Nantes (ECN) in France which consists of 266 computed nodes. Each node provides 24 cores (dual processors Intel\(\text{R}\) Xeon\(\text{R}\) CPU E5-2680 v3 @ 2.50GHz). The 2D wave simulations are performed in parallel using 24 cores. This choice is arbitrary and perhaps slightly excessive considering the fact that the 2D mesh has only about 28000 cells in total. The number of outer and inner iterations in the PIMPLE algorithm is fixed to 5 and 4, respectively; and the iterative non-orthogonal correction is set to 1. Hence, at each time step, the flow solver solves the predictive momentum equations 5 times and the Poisson’s equation for the dynamic pressure 20 times. On 24 cores each time step in the 2D wave simulations is completed within 0.4 to 0.5 seconds real time. The 3D seakeeping simulations (total cell count \(\approx 1.8\) millions) use 72 cores with 20 outer iterations, 3 inner iterations and 1 iterative non-orthogonal correction. As a result 20 predictive momentum equations and 120 Poisson’s equations are solved at each time step. The time required to complete one time step in the 3D simulations using the selected 72 cores varies between 38 and 43 seconds real time. The computed nodes spend a large amount of the CPU time to recompute the geometrical properties of each cell and to deform the 3D mesh according to the rigid-body motion of the hull. It is noted that most likely the performance can be improved significantly by selecting a more efficient technique to handle body motions and dynamic meshing in the computational domain.

The wave condition is considered nonlinear and the stream function wave model has been selected as the input wave model at the wave inlet boundary. At the initial time the volume fraction field \(\alpha\) and the velocity field \(u\) are initialized according to this model. Three numerical wave probes are defined in the 2D domain at AP (aft perpendicular), MS (midship) and FP (forward perpendicular). These numerical wave probes compute the free surface elevation from the volume fraction field by assuming that the free surface is at \(\alpha = 0.5\). While this assumption allows a simple and efficient reading of the free surface elevation it must be stressed that the probed values are dependent on the local grid resolution at the probe locations. A primitive test shows that on the meshes used in in the current work the errors in the numerical wave probes affect the resulting 1st-harmonic (normalized by the analytical equivalent) in the order of \(O(10^{-3})\).

4.1 Wave propagations in the 2D domain

The free surface elevation at midship after more than \(40T_e\) (encountered wave period) is shown in Fig. 3 (left). The crest and trough are not symmetric which indicates that the wave condition is slightly nonlinear. The time step is kept constant at 0.01 s which is equivalent to \(T_e/\Delta t \approx 187\). The simulation become unstable very quickly when using the standard Crank-Nicolson scheme (i.e. CrankNicolson 1, see Table 2). After a few \(T_e\) the Courant number grows from 0.7 to more than 200 and the simulation crashes at about \(4.5T_e\). While perhaps the
stability can be restored by reducing the time step significantly we have not explored further this option since it may quickly become very resource demanding to reduce the time step much further. An alternative option is to reduce the off-centering coefficient \(c_o\). Here at \(c_o = 0.9\) (i.e. CrankNicolson \(0.9\)) the simulation runs stable for more 40 encountered period. It is possible to lower the \(c_o\) value even further at the cost of additional lost of the wave amplitude as shown in Fig. 4 (left). When the time step is fixed at 0.01 s (\(T_e/\Delta t \approx 187\)) there is a lost of about 4% at midship at \(c_o = 0.7\) against 2% at \(c_o = 0.9\). Further improvements can be observed when reducing the time step as shown in Fig. 4.

Figure 3 (right) shows 1\textsuperscript{st} harmonics of the free surface elevations. These 1\textsuperscript{st} harmonics are the results of moving FFTs applied with a window size of 1\(T_e\). Here it is evident that when using the CrankNicolson scheme the resulting the free surface elevations \(\eta\) are very similar at FP, midship and AP. Hence the the wave amplitude does not show sign of a significant decay when it is propagated numerically from FP to AP. The Euler scheme too is stable but the wave amplitudes are very different between FP, midship and AP. Indeed waves are decaying strongly over the ship length (\(L_{pp} \approx 0.5\lambda\)). With the Euler scheme the incident wave has lost more than 12% of its 1\textsuperscript{st} harmonic amplitude at FP and 19% by the time it reaches AP. Since there is only about 0.5\(\lambda\) from FP to AP the decay rate is almost 13% per \(\lambda\) which is very significant in most engineering applications. Figure 4 (right) shows the temporal convergence rate for the Euler scheme compared to the modified CrankNicolson scheme with the off-centering coefficient \(c_o\) varied between 0.7 and 0.9. For these the temporal convergence tests the 2D mesh is kept the same and so are the spatial discretization schemes as shown in Table 2. As expected the Euler scheme is (approximately) 1\textsuperscript{st}-order. The CrankNicolson schemes at \(c_o = 0.7\) and \(c_o = 0.8\) are confirmed here to be 1\textsuperscript{st}-order also but the level of errors are one order of magnitude lower than the Euler scheme. Theoretically the CrankNicolson scheme is fully 2\textsuperscript{nd}-order accurate only when \(c_o = 1\). Since the scheme currently is not stable at \(c_o = 1\) the closest results
are with $c_o = 0.9$ where the errors seem to indicate a convergence rate of close-to 2nd-order ($\approx 1.7$). Nonetheless these results must be evaluated taking into account the previously discussed limitations of the numerical wave probes used to determine the free surface elevation from the volume fraction field. The numerical wave probes produce errors in the order of $O(10^{-3})$ which is the same order as the errors shown for $c_o = 0.9$ in Fig. 4 (right). Hence the errors seen here at $c_o = 0.9$ may very well contain a relatively significant contribution from the errors in the numerical wave probes.

4.2 Seakeeping simulations in regular head sea conditions

When using the Euler scheme the seakeeping simulations run stable for more than $17T_e$. The 2D wave propagation tests show clearly that the Euler scheme is not appropriated since waves are decaying significantly from FP to AP. The 3D seakeeping mesh has the similar mesh configuration as in 2D and all relevant numerical schemes as well as the time step are kept the same between 2D and 3D simulations. Hence it is expected that the resulting incident waves when using the Euler scheme in 3D have the same decay rate as seen on the 2D mesh. When inspecting the resulting heave and pitch motions, however, it is not clear how the condition where waves at FP and AP may have different amplitudes affects the heave and pitch motions. Figures 6 and 7 show results of a moving FFT on heave and pitch motions. Since the simulated incident waves are decaying and therefore have some loss of amplitude the resulting heave and pitch motions are reduced accordingly. The resulting ship motions are expected to be improved when using the CrankNicolson scheme.

With CrankNicolson 0.9 the 2D wave simulation on the 2D mesh is stable. However, the 3D seakeeping mesh has small irregular cells which are produced in the process of cutting and fitting the mesh to the ship hull. These irregular cells are expected to have a negative influence on the overall numerical stability. Furthermore the presence of the ship hull and its motion induces a larger local flow in the vicinity of the hull which causes the local Courant number to
increase. It is our discovery that the 3D seakeeping simulation at \( c_o = 0.9 \) becomes unstable very quickly. Once again the stability can be restored by reducing the value of \( c_o \) further toward 0 (i.e. the Euler scheme) and at same time decrease the time step (see Fig. 4) considerably to maintain an acceptable accuracy of wave simulations. In the current 3D seakeeping case we are exploring a more efficient solution by reducing \( c_o \) not globally but in locally selected cells where the local Courant numbers either are considered too large or have poor geometrical properties. The selection is done based on authors own intuition and convenience rather than on a rigorous optimization. Figure 5 illustrates these local cells (shaded region). The selection is done manually prior to each run using cell selection tools available in OpenFOAM. Here a distance based cell selection tool has been used to selected all cells within a normal distance of 0.0264 m to the ship hull. This short distance covers a few layers of cells which, in some regions, can be very small. The simulation is performed using CrankNicolson 0.9 in all cells except those selected for used with CrankNicolson 0 (i.e. equivalent to the implicit backward Euler scheme, see also Eq. (7)). With this selection the off-centering coefficient \( c_o \) changes abruptly from 0.9 to 0. While it is possible to distribute the \( c_o \) values more smoothly no such attempt has been made since the current selection is already very stable. The seakeeping simulations run for more than 17\( T_e \). The resulting heave and pitch motions are shown in Fig. 6 and 7 with label CN0.9+Euler. The time series of heave and pitch motions are processed in to the frequency domain using a moving FFT with a window size of 1\( T_e \). The evolution of the 1st harmonic confirms that the resulting motions have reached a quasi-steady state in less than 17\( T_e \). More importantly the predicted amplitude of the 1st harmonics (for both heave and pitch) are significantly closer to the experiments. Some small variations still exist, possibly due to disturbance produced by a non-perfect wave treatments at the outlet boundaries. Table 3 shows a comparison of the amplitudes of the 1st harmonics between the current simulations and other CFD codes reported in [12]. Using the experiments as a reference for the comparison it is evident that the CN0.9+Euler scheme is superior than the Euler scheme. With respect to results from other CFD codes the percentage errors in the present simulations using CN0.9+Euler are approximately at the same level as reported by UNIZAG-FSB/swenseFoam and FORCE/StarCCM+. While these results are very encouraging for a further study of the CN0.9+Euler scheme it shall be stressed that the use of the CN0.9+Euler scheme combination is not strictly unambiguous.
due to the uncertainty on how to select properly the distribution of $c_o$. Moreover there are other temporal schemes which may perhaps produce results at least at a similar level of accuracy and performance without introducing ambiguous parameters as with the CN0.9+Euler scheme. The major barrier in OpenFOAM, however, is that MULES which is the dominating solver for the transport equation of the phase fraction field currently only supports the Euler and CrankNicolson schemes for the reasons that these two schemes can keep the $\alpha$-field bounded between 0 and 1.

5 CONCLUSIONS

The results of the 2D wave propagation tests and the 3D seakeeping tests show that OpenFOAM has the capacity to simulate regular head wave conditions reasonably well. Since the test condition presented in this paper only covers one wave condition there is a need for more tests on a larger variety of wave and test conditions in order to draw a firm conclusion on whether or not OpenFOAM can be applied satisfactory in a practical engineering application. Already the results presented here reveal some weaknesses and limitations related to the stability of the available 2\textsuperscript{nd}-order CrankNicolson scheme and the accuracy of wave probing tools. The modified CrankNicolson scheme provides a better accuracy at an increasing value of $c_o$, Fig. 4. When $c_o$ is too large, however, the simulation may become unstable. Here it is shown that the stability can be restored by reducing $c_o$ value only at local regions where the cell quality or the flow are considered critical. Without reducing the $c_o$ value globally waves are allowed to propagate more accurately toward the ship which helps to improve significantly the overall accuracy of the predicted ship motions.
Figure 7: Normalized pitch motion (left) and the corresponding results of the moving FFT (right) using a window size of $1T_e$. Pitch is normalized by $k\xi$, where $k$ is the wave number and $\xi$ is the wave amplitude $\xi = H/2$. The time step size is $T_e/\Delta t \approx 187$. 
Table 1: The seakeeping case used in this work c.f. [12]. Wave crest is at FP at \( t = 0 \) s. Heave motion normalized by wave amplitude \( \xi = H/2 \), pitch angle by \( k\xi \), forces by \( 0.5\rho U^2 S_0 \). Sinkage is positive upwards. Trim is positive bow up.

<table>
<thead>
<tr>
<th>Hull data (model scale)</th>
<th>Length btw. perpendiculars, Lpp, [m]</th>
<th>6.0702</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Maximum beam of waterline, B [m]</td>
<td>0.8498</td>
</tr>
<tr>
<td></td>
<td>Draft, [m]</td>
<td>0.2850</td>
</tr>
<tr>
<td></td>
<td>Displacement volume, [m³]</td>
<td>0.9571</td>
</tr>
<tr>
<td></td>
<td>LCB, [%Lpp, fwd+]</td>
<td>-1.4800</td>
</tr>
<tr>
<td></td>
<td>Vert. CoG, [m, from keel]</td>
<td>0.3780</td>
</tr>
<tr>
<td></td>
<td>Moment of Inertia, [Kxx/B]</td>
<td>0.4000</td>
</tr>
<tr>
<td></td>
<td>Moment of Inertia, [Kyy/Lpp, Kzz/Lpp]</td>
<td>0.2520</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Test condition</th>
<th>Degree-of-freedom,</th>
<th>heave &amp; pitch</th>
</tr>
</thead>
<tbody>
<tr>
<td>(Towed in regular head wave)</td>
<td>Towing speed, [m/s]</td>
<td>2.0170</td>
</tr>
<tr>
<td></td>
<td>Wave length, ( \lambda ) [m]</td>
<td>11.840</td>
</tr>
<tr>
<td></td>
<td>Wave height, H [m]</td>
<td>0.1960</td>
</tr>
<tr>
<td></td>
<td>Froude number, ( F_r )</td>
<td>0.2610</td>
</tr>
<tr>
<td></td>
<td>Reynolds number, ( R_e )</td>
<td>( 1.074 \cdot 10^7 )</td>
</tr>
<tr>
<td></td>
<td>Water depth, ( d ) [m]</td>
<td>5.5</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Derived wave properties based on [5]</th>
<th>Period, ( T ) [s]</th>
<th>2.7581</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Wave frequency, ( \omega ) [rad/s]</td>
<td>2.2781</td>
</tr>
<tr>
<td></td>
<td>Encountering frequency, ( \omega_e ) [rad/s]</td>
<td>3.3484</td>
</tr>
<tr>
<td></td>
<td>Wave number, ( k ) [1/m]</td>
<td>0.5307</td>
</tr>
<tr>
<td></td>
<td>( kd ) [-]</td>
<td>2.9186</td>
</tr>
<tr>
<td></td>
<td>( H/\lambda ) [-]</td>
<td>0.0165</td>
</tr>
<tr>
<td></td>
<td>Phase velocity, ( c ) [m/s]</td>
<td>4.2929</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Experimental results c.f. [12] ampl.</th>
<th>Wave, ampl. ([/Lpp]) 1(^{st})-harm. ( 0.01611 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>ampl. ((0^\text{th}, 1^\text{st}, 2^\text{nd}, 3^\text{rd}, 4^\text{th}))</td>
<td>Thrust Coeff., CT, ampl. ([10^4]), phase [rad]</td>
</tr>
<tr>
<td>phase ((-0.3585, 0.0890, 1.5306, -0.8437))</td>
<td>ampl. ((10.842, 25.101, 0.655, 0.581, 0.595))</td>
</tr>
<tr>
<td>phase ((-0.2005, 0.9312, 0.0206, 0.0013, 0.0002))</td>
<td>phase ((-1.7205, 1.3007, 1.6636, 0.5488))</td>
</tr>
<tr>
<td>Pitch, ampl. ([/k\xi]), phase [rad]</td>
<td>ampl. ((-0.0562, 1.1185, 0.0379, 0.0017, 0.0003))</td>
</tr>
<tr>
<td>phase ((-0.5811, 2.1387, 0.8706, -0.3979))</td>
<td>phase ((-0.5811, 2.1387, 0.8706, -0.3979))</td>
</tr>
</tbody>
</table>
### Table 2: The description of the FVM schemes used in this work

<table>
<thead>
<tr>
<th>FVM Scheme</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Euler</td>
<td>1st-order implicit backward Euler scheme</td>
</tr>
<tr>
<td>CrankNicolson $c_0$</td>
<td>Modified Crank-Nicolson with off-center coefficient $c_0$</td>
</tr>
<tr>
<td>backward</td>
<td>2nd-order three-time-steps backward differencing scheme</td>
</tr>
<tr>
<td>vanLeer01</td>
<td>2nd-order scheme with van Leer’s limiter bounded between [0, 1]</td>
</tr>
<tr>
<td>interfaceCompression</td>
<td>High-resolution scheme for the artificial compression term</td>
</tr>
<tr>
<td>linear</td>
<td>2nd-order central differencing scheme (linear interpolation)</td>
</tr>
<tr>
<td>linear correct</td>
<td>Linear interpolation with explicit non-orthogonal correction</td>
</tr>
<tr>
<td>linearUpwind</td>
<td>2nd-order upwind scheme</td>
</tr>
</tbody>
</table>

### Table 3: Amplitudes of the 1st harmonic of normalized heave ($z/\xi$) and pitch ($\theta/(k\xi)$) motions c.f. [12]. The experimental data (EFD) is from the towing tests performed at FORCE Technology. The errors in percent are computed as $100(S-EFD)/EFD$, where $S$ are the simulated results and EFD are the experiments.

<table>
<thead>
<tr>
<th>KCS CASE 2.10-C5</th>
<th>heave, $z/\xi$</th>
<th>Err. [%]</th>
<th>pitch, $\theta/(k\xi)$</th>
<th>Err. [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>EFD</td>
<td>0.9312</td>
<td>-</td>
<td>1.1185</td>
<td>-</td>
</tr>
<tr>
<td>Present Simulations/CN0.9+Euler</td>
<td>0.866</td>
<td>-7.00</td>
<td>1.050</td>
<td>-6.12</td>
</tr>
<tr>
<td>Present Simulations/Euler</td>
<td>0.808</td>
<td>-13.23</td>
<td>0.962</td>
<td>-13.99</td>
</tr>
<tr>
<td>FORCE/StarCCM+</td>
<td>0.874</td>
<td>-6.14</td>
<td>1.084</td>
<td>-3.08</td>
</tr>
<tr>
<td>HSVA/FreSCO+</td>
<td>0.899</td>
<td>-3.46</td>
<td>1.087</td>
<td>-2.81</td>
</tr>
<tr>
<td>IIHR/CFDShip-Iowa v.4.5</td>
<td>0.917</td>
<td>-1.52</td>
<td>1.090</td>
<td>-2.55</td>
</tr>
<tr>
<td>KRISO/WAVIS</td>
<td>0.741</td>
<td>-20.43</td>
<td>0.886</td>
<td>-20.79</td>
</tr>
<tr>
<td>MARIC/FINEMarine313</td>
<td>0.944</td>
<td>1.37</td>
<td>1.163</td>
<td>3.98</td>
</tr>
<tr>
<td>NMRI/NAGISA</td>
<td>0.925</td>
<td>-0.67</td>
<td>1.047</td>
<td>-6.39</td>
</tr>
<tr>
<td>NUMECA/ISISCFD</td>
<td>0.898</td>
<td>-3.57</td>
<td>1.080</td>
<td>-3.44</td>
</tr>
<tr>
<td>UM/OF23x</td>
<td>0.901</td>
<td>-3.24</td>
<td>1.078</td>
<td>-3.62</td>
</tr>
<tr>
<td>UNIZAG-FSB/swenseFoam, Grid#1</td>
<td>0.876</td>
<td>-5.93</td>
<td>1.038</td>
<td>-7.20</td>
</tr>
</tbody>
</table>
REFERENCES


SIMULATION OF FLOATING BODIES USING A COMBINED IMMERSED BOUNDARY WITH THE LEVEL SET METHOD IN REEF3D

HANS BIHS*, ARUN KAMATH*, JIAYI ZHENG LU* AND ØIVIND A. ARNTSEN*

*Department of Civil and Environmental Engineering
NTNU Trondheim
Høgskolereingen 7A, 7491 Trondheim, Norway
e-mail: hans.bihs@ntnu.no, web page: http://www.reef3d.com/

Key words: CFD, REEF3D, 6DOF, floating body, free surface flow, level set method

Abstract. The 6DOF algorithm implemented in the open-source computational fluid dynamics (CFD) model REEF3D is used to simulate a horizontal cylinder in heave motion around the free surface and the motion of a freely floating rectangular barge in waves. The numerical model uses a staggered Cartesian grid. The free surface is obtained with the level set method. The floating body is described with a primitive triangular surface mesh neglecting connectivity. A ray-tracing algorithm is used to determine the intersection of this surface mesh with the underlying Cartesian grid. In this way, the model avoids re-meshing the domain while calculating the motion of the floating object. The moving fluid-solid boundary is treated with the immersed boundary ghost cell method. In order to validate the model, two test cases are presented. First, the damping of the heave motion after providing an initial excitation by raising the center of gravity of the cylinder slightly above the free surface is calculated and the numerical results are compared to experimental data. Then, the motion of a freely floating rectangular barge is simulated under different wave conditions. The barge is free to move in surge, roll and heave motions. The numerical results are compared to the experimental results to validate the algorithm. The results show that the numerical model REEF3D represents the motion of floating objects well, with good agreement with the experimental results.

1 INTRODUCTION

A large number of engineering problems in various fields involve fluid-structure interaction. In the field of marine and coastal engineering, the fluid structure problem is further challenging due to the presence of the free surface. The free surface interaction plays an important role in the hydrodynamics of a floating structure. With several important economic activities in the domain of marine engineering, it is essential to develop a better understanding of the interaction of the floating bodies on water such as ships, floating piers, floating breakwaters or floating ice floes. Such investigations have been traditionally carried out using physical experiments. With
the advances in computational modelling, deep insights into the fluid-floating body hydrodynamics can be obtained through numerical modelling. Here, solving the Navier-Stokes equations can calculate complex physical processes involved in the interaction problem such as the viscous effects from turbulence, complex free surface phenomena due to the impact of an object and non-linear effects on the body due to non-linear wave events.

An earlier approach to fluid-structure interaction Navier-Stokes solvers has been to use Arbitrary Lagrangian-Eulerian (ALE) methods [17], [21]. When the structure is moving, the underlying mesh is re-adjusted in order to track the movement of the solid. For complex flow situations as often encountered in the field of marine engineering, this can limit the applicability of the algorithm. More flexibility is achieved, when two individual grids are used, one for the underlying fluid flow and another for the moving solid. The overset mesh implementation (e.g. [5]) avoids re-meshing at the cost of continuously re-mapping the overset region. Yang et al. [22] presented a two-phase flow solver with a sharp interface immersed boundary method for embedding the moving solids in the fluid flow. This way, the grid remains fixed and avoids re-meshing as well as the operations needed for the overset mesh approach.

In the current study the 6DOF algorithm in REEF3D is present. REEF3D is an open-source CFD code, developed at the Department of Civil and Environmental Engineering at Norwegian University of Science and Technology (NTNU), Norway. The software has been successfully used for several marine applications, such as wave forces and free surface analysis around horizontal cylinders in tandem [15], waves forces and wave elevation around vertical cylinders [13] or breaking wave forces [4]. At first, the model is validated with a free heaving of a circular disk case. Then, the free floating of a rectangular box is investigated for wave conditions.

2 NUMERICAL MODEL

The numerical simulations in the present paper are performed with the open-source CFD code REEF3D [3]. For the hydrodynamics of the flow, the continuity equation and the incompressible Reynolds-Averaged Navier-Stokes equations are solved:

\[
\frac{\partial u_i}{\partial x_i} = 0
\]  

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_t) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + g_i
\]

where \( u \) is the velocity \( t \), \( \rho \) is the fluid density, \( p \) is the pressure, \( \nu \) is the kinematic viscosity, \( \nu_t \) is the eddy viscosity and \( g \) the acceleration due to gravity. For turbulence, the two-equation \( k-\omega \) model is selected. Additional limiters for the eddy-viscosity [8] and turbulence at the free surface [14] are included to avoid turbulence over-production typical for oscillatory two-phase flow, see [3] for details. The convective terms of the Navier-Stokes equation are treated with the fifth-order accurate conservative WENO (weighted essentially non-oscillatory) scheme in a finite difference framework [12]. The second-order TVD Runge-Kutta scheme is used for the temporal discretization [19], while the time step is selected adaptively based on the CFL number [7]. The transport equations of the turbulence model are solved implicitly, in order to avoid the time step restrictions arising from the large source terms for turbulence production and dissipation. The pressure is treated with the projection method [6] and the resulting Poisson equation is solved.
using HYRPE’s geometric multigrid PFMG [1] preconditioned BiCGStab solver [9]. The free water surface aspect of the flow is considered through a two-phase flow approach. For the interface capturing, the level set method is used [16]. The free surface is then the zero-contour of the signed distance function $\phi$ with the following properties:

$$
\phi(\vec{x}, t) = \begin{cases} 
> 0 & \text{if } \vec{x} \in \text{phase 1} \\
0 & \text{if } \vec{x} \in \Gamma \\
< 0 & \text{if } \vec{x} \in \text{phase 2}
\end{cases}
$$

(3)

The level set function is propagated with a simple convection equation, where the Hamilton-Jacobi version of the WENO scheme [11] is used for spatial discretization:

$$
\frac{\partial \phi}{\partial t} + u_j \frac{\partial \phi}{\partial x_j} = 0
$$

(4)

Under the flow propagation, the level set function tends to loose its signed distance property, making re-distancing necessary. The current approach uses the PDE based level set reinitialization [20]:

$$
\frac{\partial \phi}{\partial t} + S(\phi) \left( \frac{\partial \phi}{\partial x_j} - 1 \right) = 0
$$

(5)

Both equations Eqs. 4 and 5 use the third-order TVD Runge-Kutta scheme [19] for advancing in time. For the simulation of the free floating box, waves are generated using the relaxation method [3].

3 6DOF ALGORITHM

An overview of the fluid-structure interaction algorithm for the floating body is presented [2]. A triangular surface mesh without connectivity is created to define the geometry of the body. A ray-tracing algorithm, calculating the shortest distance to the closest triangle for the grid point, is used to determine the intersections of the surface mesh with the underlying Cartesian grid. The level set method is used to define the solid boundary of the floating object. The standard reinitialization algorithm is used to obtain signed distance properties for the level set function in the vicinity of the solid body. The forces on the surface $\Omega$ are determined for each direction $i$ separately with the pressure $p$ and the viscous stress tensor $\tau$:

$$
F_{i,e} = \int_{\Omega} (-n_ip + n_i \cdot \tau)d\Omega
$$

(6)

The moments around the center of gravity can be calculated with the following equation:

$$
L_{i,cg} = \int_{\Omega} r_{cg} \times (n_ip + n_i \tau)d\Omega
$$

(7)

As the outer geometry of the floating body is represented through a level set function, the discrete surface area can be determined with a Dirac delta function:

$$
d\Omega = \int \delta(\phi)|\nabla \phi|dx
$$

(8)
The dynamic rigid body equations can be solved based on the forces \( X, Y, Z \), the moments \( K, M, N \) and the moments of inertia \( I_x, I_y, I_z \):

\[
F_i = J_i^{-1} F_{i,e} = [X, Y, Z] \\
L_i = J_i^{-1} L_{i,e} = [K, M, N]
\]  

(9)

with the equations of motions for the six degrees of freedom:

\[
[m(\ddot{u} - \nu r + \omega q)] = X \\
[m(\ddot{v} - \omega r + \nu q)] = Y \\
[m(\ddot{w} - \nu r + \omega q)] = Z \\
[m(\ddot{\omega} - \nu r + \omega q)] = Z \\
[I_x(\dot{p} + (I_z - I_y)qr)] = K \\
[I_y(\dot{q} + (I_x - I_z)rp)] = M \\
[I_z(\dot{r} + (I_y - I_x)pq)] = N
\]  

(10)

where \( u, v, w \) are the linear velocities and \( p, q, r \) the angular velocities. Then \( \dot{\omega}, \dot{\nu}, \dot{\omega}, \dot{\nu} \) and \( \dot{\nu} \) can be calculated in an explicit manner.

The generic linear or angular velocities \( \dot{\varphi} \) and the generic position or orientation vector \( \varphi \) of the floating body can be calculated with a second-order Adams-Bashforth scheme for the new time-step:

\[
\dot{\varphi}^{n+1} = \dot{\varphi}^t + \frac{\Delta t}{2} (3\ddot{\varphi}^{n+1} - \ddot{\varphi}^n) \\
\varphi^{n+1} = \varphi^n + \frac{\Delta t}{2} (3\dddot{\varphi}^{n+1} - \dddot{\varphi}^n)
\]  

(11)

The dynamic rigid body equations are solved in the floating body reference frame, while the fluid forces and moments are calculated in the earth-fixed reference frame. As a result, coordinated transformations in both directions are necessary [5][2].

4 RESULTS

4.1 Heaving of a Circular Cylinder

For validation purposes, a 2D free heaving body is simulated. The movement of the body is limited to the vertical degree of freedom and it is allowed to oscillate freely. Experimental studies has been performed by Ito [10] with a circular cylinder and the data will be used for comparison with the numerical model. The spatial domain of the numerical wave tank (NWT) is 10m long and 1.8m high. The water depth is \( d=1.22 \)m and the water is initially still. The density of the water is \( \rho=1000 \)kg/m\(^3\).
The configuration of the body position follows Ito’s [10] set up where the object is located in the centre of the tank. The body is a horizontal circular cylinder with a radius $r=0.0762m$ and half of water density, $\rho=500kg/m^3$. The cylinder is partially submerged with its centroid at $h_1=0.02454m$ above the free surface. The cylinder is released from this initial position in a free fall and enters the water. A uniform grid in the domain is used for the simulations, there are three different mesh sizes used: 0.025m, 0.02m and 0.01m with a grid size of 400x72, 500x90 and 2000x180 in the horizontal and vertical directions, respectively.

Results presented in Fig. 1 have a normalized Z-axis subtracting the water depth and dividing by the initial displacement $h_i$ while the X-axis represents time.

\[
Normalization = \frac{Z - d}{h_i}
\]  

In Fig. 1, the mesh size of 0.025m approximates the experiment in the first oscillation but diverges from there, having a high oscillation after 1.5s. The 0.02m grid size is closer to the experiment data, however the oscillations start to differ after 1s. The mesh size of 0.01m matches the experimental data, representing the behaviour of the cylinder correctly. After 1.5s the position start to differ slightly, due to the effect of the reflected waves generated from the left and right side walls. The mesh size of 0.01m will be used for additional analysis. Snapshots in 0.5s time steps until $t=4.0s$ are shown in Fig. 2. The cylinder breaks the water surface, creating a series of waves to the sides that dissipates over time and length of the tank. It can be seen that the highest velocity occurs at $t=0.5s$ when the cylinder enter and breaks the still water surface. After that, waves are created at both sides, disappearing at $t=3.0s$. The release height is not enough to produce a total immersion of the cylinder which would create a water jet and separation of the free water surface. The water surface matches well with numerical simulations carried out by Yang [22]. The vorticity is shown in Fig. 3 with an horizontal line representing the water level and the boundary between air and water.
Figure 2: Cylinder velocity with $t=0.5s$ time-step

Figure 3: Vorticity around the free heaving circular disc
4.2 Free Floating Box in Waves

A floating rigid body is simulated in 2D for wave conditions. The degrees of freedom in the simulation are the movement along the X and Z axis and the rotation around Y axis. The floating body consists of a box or rectangular barge with a length of $l=0.30\text{m}$ and a height of $h=0.20\text{m}$ following the experiment realised at Dalian University by Ren et al. [18]. The water depth is $d=0.4\text{m}$. The density of the floating body is $\rho=500\text{kg/m}^3$. The results will be compared with the experiment.

The simulation is performed in a 20m long and 1.5m high tank. The tank is divided in three zones where the initial wave zone is equivalent to one wavelength and the end zone or numerical beach is equal to two wavelengths, for wave generation and absorption, respectively. For the simulation, the box is positioned at 7m from the start of the tank so the wave reflection does not affect the simulation results. The comparison will be done after some time when the series of waves stabilize and motion starts. For the waves, 2nd-order Stokes theory is used, with a wave period $T=1.2\text{s}$, a wave height $H=0.04\text{m}$, a wavenumber $k=3.245\text{m}^{-1}$ and a wavelength of $L=1.936\text{m}$. Three different grids are used in the case: 0.025m, 0.01m and 0.005m with a total cells of 800x60, 2000x150 and 4000x300, respectively. The wave elevation is presented in 4, it shows a good agreement between numerical results and experimental data. The only difference occurs in the trough of the third and fifth wave which is slightly lower than model results.

![Figure 4: Wave elevation](image)

The numerical heave results in Fig. 5a match the experimental measurements well. The results are accurate in relation to the height except the crests between the first and second waves. All the grids show a good match with the experiment data. In Fig. 5b a detail of the matching part is showed with only the 0.005 grid.
(a) Heave motion with three grid sizes and experiment data

(b) Heave motion between $t/T=6.0$ and $t/T=12.0$

(c) Roll motion with three grid sizes and experiment data

(d) Roll motion between $t/T=6.0$ and $t/T=12.0$

(e) Surge motion with three grid sizes and experiment data

(f) Surge motion between $t/T=6.0$ and $t/T=12.0$

Figure 5: Movement of the free floating box
In Figs. 5c and 5d, the roll motion of the box is presented. The simulated roll peaks almost coincide with the experimental observations. It becomes clear from Fig. 5c, that the coarsest grid with $dx=0.025\text{m}$ gives unsuitable results, while the two finer meshes perform well. Similar to the other degrees of freedom, the two finer meshes successfully capture the surge motion of the free floating box (Fig. 5e). Fig. 6 shows the contour of the vertical velocity around the floating box at different points in time of the wave.

![Figure 6: Rectangular barge vertical velocity in $t/T=0.5$ intervals](image)

5 CONCLUSIONS

The floating body algorithm implemented in REEF3D has been presented, validated and tested for two different cases: a heaving cylinder and a free floating rectangular barge. The cases have been studied with different grid sizes. It has been noted that a grid size of 0.01m presents remarkably good results. For the heaving cylinder, the oscillations in the position of the cylinder match correctly the experimental data. The free floating box matches very well with the experimental data for all the motions. The present study has shown that REEF3D can be used for simulations of floating bodies. However, more studies are needed in relation to floating bodies. The next step would be simulations of floating bodies in a wave field in three dimension to validate all six degrees of freedoms for wave-body interaction. Many structures are moored and the motion of a moored floating body in waves will be targeted in future research.
ACKNOWLEDGEMENT

This research was supported in part with computational resources at NTNU provided by The Norwegian Metacenter for Computational Sciences (NOTUR), http://www.notur.no.

References


Study on the Rudder Characteristics of Ultimate Rudder by Numerical Calculation

by Yasuhiro Tendou, Yoshihisa Okada, Taro Kajihama, Toshie Kajino, Kenta Katayama, Masafumi Okazaki, Kenichi Fukuda

Propeller design department, NAKASHIMA PROPELLER Co.,Ltd.
688-1, Joto-Kitagata, Higashi-ku, Okayama 709-0625, Japan

E-mail: y-tendou@nakashima.co.jp, yoshihisa@nakashima.co.jp
    t-kajihama@nakashima.cp.jp, t-kajino@nakashima.co.jp
    ken-katayama@nakashima.co.jp, m-okazaki@nakashima.co.jp
    k-fukuda@nakashima.co.jp

Web page: http://www.nakashima.co.jp

Key words: Ultimate Rudder, Rudder with bulb, Rudder characteristics, CFD

Abstract. The authors invented Ultimate Rudder as the rudder with bulb. The authors calculated the rudder characteristics of the normal rudder and Ultimate Rudder by CFD at several steering angles and compared these values. The result showed that regardless of the presence or absence of the bulb, signs of separation appear on the control surface with a steering angle of 20 deg. to 30 deg. and regarding the steering torque coefficient, it was found that the steering torque coefficient of Ultimate Rudder is larger than the normal Rudder when the steering angle is less than 20 deg. and also the steering torque coefficient can be decreased by changing the shape of the rudder bulb.

1 INTRODUCTION

Since EEDI was applied, the demand of Energy Saving Devices (ESD) has been increased. NAKASHIMA PROPELLER developed ECO-Cap as the cap with fin and Ultimate Rudder as the rudder with bulb so far. In particular, the rudder with bulb at fore side of rudder has a long history and has been equipped on many vessels. Many studies about propulsive performance of rudder with bulb have been carried out. On the other hand, steering performance is also required for rudder, but there are not many studies about the steering performance of the rudder with bulb. The rudder characteristics are defined by lift-drag characteristics at each angle. In this paper, the rudder characteristics of the rudder both with and without bulb at aft side of propeller were calculated by CFD.

2 CALCULATION BY CFD

2 rudders were compared in this paper. One is a normal rudder without bulb. Another is Ultimate Rudder. And then these rudders and 6 blades propeller were designed for mega container vessel. The propeller particulars are shown in Table 1 and over views of rudders are shown in Fig.1.

The scale ratio is 39.58. The chord length of rudder center is 163.4mm, the height is
The clearance between bulb and cap is 2.5mm in model scale. These compared rudders have same rudder section.

In this paper, the authors calculated rudder characteristics by SOFTWARE CRADLE “SCRYU/Tetra Ver.10” which is commercial CFD code. The number of mesh was about 21 million. k-kl-ω model was applied for turbulent model. The analyzed model is shown in Fig.2. The rudder is settled after the propeller same as actual vessel. This calculation was analyzed in model scale. The propeller was rotated in 8rps. 1.44m/s non-uniform wake flow was given at inlet boundary. Lift, drag, and steering torque coefficients were calculated at each rudder angle, and these were compared between normal rudder and Ultimate Rudder. Fig.3 shows the coordinate system.

<table>
<thead>
<tr>
<th>Table 1: Principal particulars of propeller</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blades</td>
</tr>
<tr>
<td>MPNO.1</td>
</tr>
<tr>
<td>Diameter (m)</td>
</tr>
<tr>
<td>0.240</td>
</tr>
<tr>
<td>Pitch ratio at 0.7R</td>
</tr>
<tr>
<td>1.0326</td>
</tr>
<tr>
<td>Expanded area ratio</td>
</tr>
<tr>
<td>0.7695</td>
</tr>
<tr>
<td>Boss ratio</td>
</tr>
<tr>
<td>0.1789</td>
</tr>
<tr>
<td>Skew angle (deg.)</td>
</tr>
<tr>
<td>34</td>
</tr>
</tbody>
</table>

Fig.1 Profile of propeller and rudder

Fig.2 Analysis region of propeller and rudder with bulb
3 CALCULATION RESULTS

3.1 Comparison of lift and drag coefficient

Fig. 4 shows lift and drag coefficients for normal rudder and Ultimate Rudder by CFD calculation.

Lift coefficient of normal rudder increases from 0 deg. to 20 deg. And it is almost same level from 20 deg. to 30 deg. In contrast, lift coefficient of Ultimate Rudder tends to increase slightly even the range between 20 deg. and 30 deg. There is little difference between normal rudder and Ultimate Rudder from 0 deg. to 20 deg., but the difference is bigger at 30 deg.

Next, Fig. 5 to 8 show the pressure distribution and the streamline at the steering center cross section of Ultimate Rudder and the normal rudder every 10 deg. from 0 deg. to 30 deg.

With both the normal rudder and Ultimate Rudder, signs of separation appear on the control surface at 20 deg. Further, at 30 deg., a large separation region can be confirmed in both rudders. About Ultimate Rudder in fig. 8, it seems that the low pressure range spreads to the starboard side of the rudder by the bulb.
In order to investigate the influence of the pressure distribution on the rudder surface on the lift and drag, the direction and strength of the pressure are obtained by multiplying the pressure of the rudder surface and the normal vector, the lift direction component is $P_L$, the drag direction component is $P_D$.

Fig. 9 and 10 show the distributions of $P_D$ and $P_L$ on the starboard side of Ultimate Rudder and the normal rudder and the limiting stream line at 30 deg. The lower value of $P_D$ means greater drag, and the higher value of $P_L$ means greater lift force.

As shown in Fig. 9, lower range of $P_D$ of the normal rudder spreads to the upper side of the rudder, but lower range of $P_D$ of Ultimate Rudder is relatively widely distributed downward from the bulb. Also, it can be seen that there is lowest point of $P_D$ in the lower part of the bulb body. Both of them tend to have a tendency for $P_D$ to be lower at the part where the limiting streamlines are disturbed. It is considered that drag was generated by separation of flow.

Regarding the lift force, as shown in Fig. 10, a high range of $P_L$ is generated near the front edge of Ultimate Rudder and the normal rudder. Especially, there is a high value of $P_L$ at the front edge of the rudder lowering the turbulence of the limiting streamline, but it was confirmed that the bulb rudder has higher $P_L$ than the normal rudder.

From the calculation result of 30 deg., it was found that lift force and drag force occurred in the bulb part with the bulb rudder, which affects the lift coefficient and drag coefficient of the entire rudder.
3.2 Comparison of steering torque coefficient

Fig. 11 shows the steering torque coefficient at each steering angle of Ultimate Rudder and the normal rudder. Regarding Ultimate Rudder, the calculation was also carried out in a shape with 20 mm clearance between the propeller cap and the bulb. Regarding the adjustment of the clearance, length of the bulb is shortened without changing the position of the rudder main body.

From Fig. 11, it was found that a large steering torque in the minus direction is generated in a range where the steering angle is small with Ultimate Rudder with 2.5 mm clearance. Although the direction is different at 30 deg., almost same steering torque coefficient is generated for Ultimate Rudder and the normal rudder.

Regarding Ultimate Rudder with 20 mm clearance, the steering torque coefficient is smaller in the whole area than in the case of 2.5 mm clearance. This is probably because the bulb is shorter and the distance between the center of the rudder shaft and the leading edge of the bulb is shorter. At the steering angle of 30 deg., the steering torque coefficient is smaller than that of the normal rudder.

From the above, it was confirmed that by changing the clearance and by shortening the total length of the propeller cap and the rudder bulb, the steering torque coefficient can be adjusted.
4. Conclusion

About the propeller, Ultimate Rudder, and the normal rudder designed for mega container vessel, authors calculated the lift, drag, and the steering torque coefficient by CFD in the model scale and these values were compared. As a result, regardless of the presence or absence of the bulb, signs of separation appear on the control surface at 20 deg. and separation regions could be confirmed at 30 deg. However it was found that Ultimate Rudder does not suddenly decrease the lift coefficient even after stalling, but conversely increases slightly. This is thought to be the effect of lift and drag on the bulb body.

Regarding the steering torque coefficient, it was found that the steering torque coefficient of Ultimate Rudder is larger than the normal Rudder when the steering angle is less than 20 deg. The steering torque generating direction is different but the values are almost same when the steering angle is 30 deg. Moreover, it was confirmed that the clearance of the propeller cap and the bulb was expanded from 2.5 mm to 20 mm, the steering torque coefficient became smaller. In other words, it can be said that the steering torque coefficient can be decreased by changing the shape of the rudder bulb.

REFERENCES

A GLOBAL APPROXIMATION TO THE GREEN FUNCTION FOR DIFFRACTION-RADIATION OF REGULAR WATER WAVES IN DEEP WATER

HUIYU WU, YI ZHU, CHAO MA, WEI LI, HUIPING FU AND FRANCIS NOBLESSE

State Key Laboratory of Ocean Engineering
Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration
School of Naval Architecture, Ocean & Civil Engineering
Shanghai Jiao Tong University, Shanghai, China
e-mail: why2277@sjtu.edu.cn

Key words: Regular Water Waves, Diffraction Radiation, Green Function, Local Flow, Global Approximation

Abstract. The Green function of the theory of diffraction radiation of time-harmonic (regular) waves by an offshore structure, or a ship at low speed, in deep water is considered. The Green function $G$ and its gradient $\nabla G$ are expressed in the usual manner as the sum of three components that correspond to the fundamental free-space singularity, a non-oscillatory local flow, and waves. Simple approximations that only involve elementary continuous functions (algebraic, exponential, logarithmic) of real arguments are given for the local flow components in $G$ and $\nabla G$. These approximations are global approximations valid within the entire flow region, rather than within complementary contiguous regions as can be found in the literature.

1 INTRODUCTION

Diffraction radiation of time-harmonic water waves by an offshore structure, or a ship at low speed, within the classical framework of linear potential flow theory and the Green function method, is routinely used to predict added-mass and wave-damping coefficients, motions, and wave loads. The Green function, which represents the velocity potential due to a pulsating source at a singular point under the free surface as is well known, is an essential element of this method. Accordingly, the Green function has been studied in a broad literature, especially for the simplest case of deep water that is considered here.

The Green function $G$ can be expressed as the sum of the fundamental free-space singularity and a flow component that accounts for free-surface effects. Moreover, this free-surface component is commonly decomposed into a wave component $W$ that represents the waves radiated by the pulsating source, and a non-oscillatory local flow component $L$. This basic decomposition into a wave and a local flow component is not unique. Indeed, three alternative decompositions and related single-integral representations of the Green function $G$ are given in Noblesse [1].
Several alternative mathematical representations and approximations of $G$ and $\nabla G$ that are well suited for numerical evaluation can be found in the literature. In particular, complementary near-field and far-field asymptotic expansions and Taylor series are given in Noblesse [1] and Telste & Noblesse [2]. Several practical approximate methods for computing $G$ and $\nabla G$ have also been given. These alternative methods include polynomial approximations within complementary contiguous flow regions, given in Newman [3, 4], Wang [5] and Zhou et al. [6], and table interpolation associated with function and coordinate transformations, given in Ponizy et al. [7]. Other useful practical methods can be found in the literature, notably in Peter & Meylan [8], Yao et al. [9], D'élia et al. [10] and Shen et al. [11].

Accuracy and efficiency are essential requirements of methods for numerically evaluating $G$ and $\nabla G$, and these important aspects are considered in the practical approximate methods listed in the foregoing. Indeed, the alternative methods proposed in these studies provide accurate and efficient methods for computing $G$ and $\nabla G$.

Numerical errors associated with potential-flow panel methods stem from several well-known sources, including: (i) discretization of the wetted hull surface of an offshore structure or a ship; i.e. the number and the type (flat or curved) of panels, (ii) approximation of the variations (piecewise constant, linear, quadratic, or higher-order) of the densities of the singularity (source, dipole) distributions over a surface panel, (iii) numerical integration of the Green function and its gradient over a panel, and (iv) numerical approximation of the Green function and its gradient.

Moreover, the Green function $G$ (as well as its gradient $\nabla G$) is given by the sum of the fundamental free-space singularity, a wave component $W$ and a non-oscillatory local flow component $L$, as was already noted. Thus, numerical errors that stem from the approximation of the local flow components in the representations of $G$ and $\nabla G$ are only one part among several sources of errors associated with panel methods. While the ideal approximations to $G$ and $\nabla G$ are highly accurate and efficient as well as very simple, this ideal goal is hard to reach in practice because accuracy, efficiency and simplicity are competing requirements.

The level of accuracy that is actually required for useful practical approximations to $G$ and $\nabla G$ therefore is a fairly complicated issue. This issue is partly considered in Wu et al. [12] for the similar theory of steady ship waves (linear potential flow around a ship hull that advances at a constant speed in calm water). Specifically, the errors due to a simple analytical approximation to the local flow component $L$ in the Green function for steady ship waves are considered in that study. This approximate local flow component $L$, given in Noblesse et al. [13], is very simple and highly efficient, but not particularly accurate. Yet, this simple relatively crude approximation to the Green function for steady ship waves is found in Wu et al. [12] to yield predictions of sinkage, trim and drag that do not differ appreciably from the predictions obtained if the Green function is computed with high accuracy. This finding suggests that highly accurate approximations to the local flow components in the Green function $G$ and its gradient $\nabla G$ for the theory of wave diffraction radiation similarly may not be necessary for practical purposes.

Simple approximations to the local flow components in the representations of $G$ and $\nabla G$ for wave diffraction radiation considered in Wu et al. [14] are given here. These approximations are based on a pragmatic hybrid approach that combines numerical approximations with near-field and far-field analytical expansions, in a manner similar to that used in Noblesse et al. [13] for the Green function of the theory of ship waves. The approximations obtained here are valid within the entire flow region, i.e. are global approximations, unlike the approximations for
complementary contiguous regions given in the literature. The approximations to the local flow components given here only involve elementary continuous functions (algebraic, exponential, logarithmic) of real arguments, and provide an efficient and particularly simple method for numerically evaluating the Green function $G$, and its gradient $\nabla G$, for diffraction radiation of time-harmonic waves in deep water. The global approximations to the local flow components in $G$ and $\nabla G$ given here are similar to, but considerably more accurate than, the approximations given in Wu et al. [15].

\section{Basic integral representations}

A Cartesian system of coordinates $X \equiv (X, Y, Z)$ is used. The $Z$ axis is vertical and points upward, and the undisturbed free surface is taken as the plane $Z = 0$. Diffraction radiation of time harmonic waves with radian frequency $\omega$ and wavelength $\lambda = 2\pi g/\omega^2$, where $g$ denotes the gravitational acceleration, is considered. Nondimensional coordinates

$$x \equiv (x, y, z) \equiv (X, Y, Z) \omega^2/g$$  

are defined.

The Green function $G(x, \tilde{x})$ corresponds to the spatial component of a nondimensional velocity potential

$$\text{Re}[G(x, \tilde{x}) e^{-i\omega T}]$$  \hspace{1cm} (2)

where $T$ denotes time. Expression (2) represents the potential of the flow created at the point $x \equiv (x, y, z \leq 0)$ by a pulsating source located at the point $\tilde{x} \equiv (\tilde{x}, \tilde{y}, \tilde{z} < 0)$, or by a flux through the free surface at the point $\tilde{x} \equiv (\tilde{x}, \tilde{y}, \tilde{z} = 0)$.

The nondimensional distances between the flow-field point $x$ and the source point $\tilde{x}$ or its mirror image $\tilde{x}_1 \equiv (\tilde{x}, \tilde{y}, -\tilde{z})$ with respect to the undisturbed free-surface plane $z = 0$ are denoted as $r$ and $d$, and are given by

$$r \equiv \sqrt{(x - \tilde{x})^2 + (y - \tilde{y})^2 + (z - \tilde{z})^2} \quad \text{and} \quad d \equiv \sqrt{(x - \tilde{x})^2 + (y - \tilde{y})^2 + (z + \tilde{z})^2}$$  \hspace{1cm} (3)

The horizontal and vertical components of the distance $d$ between the points $x$ and $\tilde{x}_1$ are given by

$$0 \leq h \equiv \sqrt{(x - \tilde{x})^2 + (y - \tilde{y})^2} \quad \text{and} \quad v \equiv z + \tilde{z} \leq 0$$  \hspace{1cm} (4)

The Green function $G$ is expressed as

$$4\pi G = -1/r + L + W$$  \hspace{1cm} (5)

where $-1/r$ is the fundamental free-space Green function, and $L$ and $W$ represent a local flow component and a wave component that account for free-surface effects. The component $L$ corresponds to a non-oscillatory local flow and the component $W$ represents circular surface waves radiated by the pulsating singularity located at the source point $\tilde{x}$. The basic decomposition (5) into a local flow and waves is non unique, as was already noted. Indeed, three alternative decompositions and related integral representations are given in Noblesse [1].

The so-called near-field integral representation in Noblesse [1] is considered here. The wave component $W$ in this representation is given by

$$W(h, v) \equiv 2\pi [\tilde{H}_0(h) - iJ_0(h)] e^v$$  \hspace{1cm} (6)
where \( \tilde{H}_0(\cdot) \) and \( J_0(\cdot) \) denote the zeroth-order Struve function and the zeroth-order Bessel function of the first kind. The corresponding local flow component \( L \) is given by

\[
L(h, v) \equiv -\frac{1}{d} - \frac{4}{\pi} \int_0^{\pi/2} \text{Re} e^M E_1(M) \, d\theta \quad \text{where} \quad M \equiv v + i h \cos \theta \tag{7}
\]

and \( E_1(\cdot) \) is the usual complex exponential integral function.

The gradient \( \nabla G \equiv (G_x, G_y, G_z) \) of the Green function \( G \) is expressed in Noblesse [1] as

\[
4\pi G_z \equiv \frac{z - \tilde{z}}{r^3} + L_z + W \quad \text{where} \quad L_z = \frac{v}{d^3} - \frac{1}{d} + L \tag{8a}
\]

\[
4\pi G_h \equiv \frac{h}{r^3} + L_h + W_h \quad \text{where} \quad L_h = \frac{h}{d^3} + L_\ast \tag{8b}
\]

\[
4\pi G_x \equiv G_h \frac{x - \tilde{x}}{h} \quad \text{and} \quad 4\pi G_y \equiv G_h \frac{y - \tilde{y}}{h} \tag{8c}
\]

The wave component \( W_h \) in (8b) is given by

\[
W_h(h, v) \equiv 2\pi [2/\pi - \tilde{H}_1(h) + i J_1(h)] e^v \tag{9}
\]

where \( \tilde{H}_1(\cdot) \) and \( J_1(\cdot) \) denote the first-order Struve function and the first-order Bessel function of the first kind. The local flow component \( L_\ast \) in (8b) is given by

\[
L_\ast(h, v) \equiv \frac{4}{\pi} \int_0^{\pi/2} \text{Im} [e^M E_1(M) - 1/M] \cos \theta \, d\theta \tag{10}
\]

where \( M \) is defined by (7).

The exponential function \( e^v \) and the Bessel and Struve functions in expressions (6) and (9) for the wave components \( W \) and \( W_h \) are infinitely differentiable. Moreover, several practical and efficient alternative approximations for the Bessel and Struve functions are given in the literature; notably in Hitchcock [16], Abramowitz & Stegun [17], Luke [18], Newman [19]. Fig.1 depicts the Struve functions \( \tilde{H}_0(h) \) and \( \tilde{H}_1(h) - 2/\pi \) and the Bessel functions \( J_0(h) \) and \( J_1(h) \) for \( 0 \leq h \leq 25 \).
Huiyu Wu, Yi Zhu, Chao Ma, Wei Li, Huiping Fu and Francis Noblesse

Figure 3: Real parts of the Green function and its vertical derivative, and related local flow and wave components, at the free surface $v = 0$ for $0 \leq h \leq 15$.

Figure 4: Real part of the horizontal derivative of the Green function and related local flow and wave components at the free surface $v = 0$ for $0 \leq h \leq 15$.

3 SPECIAL CASES

In the special case $h = 0$, expressions (6) and (9) for the wave components $W$ and $W_h$ become

$$W(h = 0, v) = -i 2\pi e^v$$

$$W_h(h = 0, v) = 4 e^v$$  \hspace{1cm} (11a)

Moreover, expression (7) yields $M = v$ and the integral representations (7) and (10) simplify as

$$L(h = 0, v) = 1/v - 2 e^v \Re E_1(v + i0)$$

$$L^*(h = 0, v) = -4 e^v$$  \hspace{1cm} (11b)

Equations (8b), (11a) and (11b) then yield $G_h = 0$ for $h = 0$, in agreement with symmetry considerations. Fig.2 depicts the functions $L(h = 0, v)$ and $\Im W(h = 0, v)$ for $-10 \leq v \leq 0$.

At the free-surface plane $v = 0$, expressions (3-6), (8) and (9) yield

$$4\pi \Re G = -1/h + L(h, v = 0) + 2\pi \bar{H}_0(h) = 4\pi \Re G_z$$  \hspace{1cm} (12a)

$$4\pi \Re G_h = 2/h^2 + L_*(h, v = 0) + 2\pi [2/\pi - \bar{H}_1(h)]$$  \hspace{1cm} (12b)

The real parts $4\pi \Re G$ and $4\pi \Re G_z$ of the Green function $G$ and its vertical derivative $G_z$ at the free surface $v = 0$, and the related local flow and wave components

$$-1/h + L(h, v = 0)$$

$$2\pi \bar{H}_0(h)$$  \hspace{1cm} (13a)

are depicted in Fig.3 for $0 \leq h \leq 15$. Similarly, the real part $4\pi \Re G_h$ of the horizontal derivative $G_h$ of the Green function at the free surface $v = 0$, and the related local flow and wave components

$$2/h^2 + L_*(h, v = 0)$$

$$2\pi [2/\pi - \bar{H}_1(h)]$$  \hspace{1cm} (13b)

are depicted in Fig.4 for $0 \leq h \leq 15$.

Fig.3 and Fig.4 show that, at the free surface $v = 0$, the local flow components dominate the wave components $W$ and $W_h$ for $0 \leq h < 1$. However, the local flow components are
significantly smaller than \( W \) and \( W_h \) for \( 15 \leq h \leq 4 \), respectively. The wave components \( W \) and \( W_h \) are infinitely differentiable and can readily be evaluated, as was already noted. Simple approximations to the local flow components \( L \) and \( L^* \) defined by the integral representations (7) and (10) are now considered.

### 4 PRACTICAL APPROXIMATIONS

The infinite flow region \( 0 \leq h < \infty, -\infty < v \leq 0 \) is mapped onto the unit square

\[
0 \leq \rho \equiv d/(1+d) \leq 1 \quad 0 \leq \beta \equiv h/d \leq 1
\]

(14a)

via the relations

\[
d = \rho/(1-\rho) \quad h = \beta d \quad v = -\sqrt{1-\beta^2} \cdot d
\]

(14b)

The related variable \( 0 \leq \alpha \leq 1 \) defined as

\[
\alpha \equiv -v/d \equiv \sqrt{1-\beta^2}
\]

(15)

is also used hereafter.

The local flow components \( L \) and \( L_s \) defined by the integral representations (7) and (10) and the related local flow components \( L_z \) and \( L_h \) are approximated as

\[
L \approx L^a \quad L_s \approx L^s_a \quad L_z \approx L^a_z \equiv \frac{v}{d^3} - \frac{1}{d} + L^a \quad L_h \approx L^a_h \equiv \frac{h}{d^3} + L^a
\]

(16)

Hereafter, \( L^a, L^s_a, L^a_z \) and \( L^a_h \) denote approximations to the local flow components \( L, L_s, L_z \) and \( L_h \), respectively.

The approximate local flow component \( L^a \) is given by

\[
L^a \equiv -\frac{1}{d} + 2 \left( \frac{d^2 - v}{1 + d^3} \right) + 2 \frac{e^v}{1 + d^3} \left( \log \frac{d - v}{2} + \gamma - 2d^2 \right) + 2\rho (1-\rho)^3 R
\]

(17a)
Huiyu Wu, Yi Zhu, Chao Ma, Wei Li, Huiping Fu and Francis Noblesse

Figure 7: Local flow components $L_z$ (lines without symbols) and $L_a z$ (lines with symbols) defined by the integral representation (7), (8a) or the related approximation (16), (17) for $0 \leq \rho \leq 1$ and $\alpha = 0, 0.2, 0.4, 0.6, 0.8, 1$.

Figure 8: Local flow components $L_h$ (lines without symbols) and $L_a h$ (lines with symbols) defined by the integral representation (8b), (10) or the related approximation (16), (18) for $0 \leq \rho \leq 1$ and $\alpha = 0, 0.2, 0.4, 0.6, 0.8, 1$.

where $\gamma = 0.577\ldots$ is Euler’s constant, $\rho$ is defined by (14a), and $R$ is defined as

$$R \equiv (1 - \beta) A - \beta B - \frac{\alpha C}{1 + 6\alpha \rho (1 - \rho)} + \beta (1 - \beta) D$$  \hspace{1cm} (17b)$$

Here, $\alpha$ and $\beta$ are defined by (15) and (14a), and the polynomials $A(\rho), B(\rho), C(\rho)$ and $D(\rho)$ in (17b) are defined as

$$A \equiv 1.21 - 13.328 \rho + 215.896 \rho^2 - 1763.96 \rho^3 + 8418.94 \rho^4 - 24314.21 \rho^5 + 42002.57 \rho^6$$
$$- 41592.9 \rho^7 + 21859 \rho^8 - 4838.6 \rho^9 \hspace{1cm} (17c)$$

$$B \equiv 0.938 + 5.373 \rho - 67.92 \rho^2 + 796.534 \rho^3 - 4780.77 \rho^4 + 17137.74 \rho^5 - 36618.81 \rho^6$$
$$+ 44894.06 \rho^7 - 29030.24 \rho^8 + 7671.22 \rho^9 \hspace{1cm} (17d)$$

$$C \equiv 1.268 - 9.747 \rho + 209.653 \rho^2 - 1397.89 \rho^3 + 5155.67 \rho^4 - 9844.35 \rho^5 + 9136.4 \rho^6$$
$$- 3272.62 \rho^7 \hspace{1cm} (17e)$$

$$D \equiv 0.632 - 40.97 \rho + 667.16 \rho^2 - 6072.07 \rho^3 + 31127.39 \rho^4 - 96293.05 \rho^5 + 181856.75 \rho^6$$
$$- 205690.43 \rho^7 + 128170.2 \rho^8 - 33744.6 \rho^9 \hspace{1cm} (17f)$$

The approximate local flow component $L_a^*$ is given by

$$L_a^* \equiv \frac{2}{1 + d^3} \left( \frac{\beta + h}{d - v} - 2\beta + 2e^d - h \right) - 4e^{-d} \left( 1 - \beta \right) \left( 1 + \frac{d}{1 + d^3} \right) + 2\rho (1 - \rho)^3 R_*$$  \hspace{1cm} (18a)$$

where $R_*$ is defined as

$$R_* \equiv \beta A_* - (1 - \alpha) B_* + \beta (1 - \beta) \rho (1 - 2\rho) C_*$$  \hspace{1cm} (18b)$$
Figure 9: Absolute error $4\pi e \equiv L - L^a$ between the local flow components $L$ and $L^a$ defined by the integral representation (7) or the related approximation (17) for $0 \leq \rho \leq 1$ and $\alpha = 0, 0.2, 0.4, 0.6, 0.8, 1$.

Figure 10: Absolute error $4\pi e^* \equiv L^* - L^a$ between the local flow components $L^*$ and $L^a$ defined by the integral representation (10) or the related approximation (18) for $0 \leq \rho \leq 1$ and $\alpha = 0, 0.2, 0.4, 0.6, 0.8, 1$.

Here, the polynomials $A_*(\rho), B_*(\rho)$ and $C_*(\rho)$ in (18b) are defined as

$$A_* = 2.948 - 24.53\rho + 249.69\rho^2 - 754.85\rho^3 - 1187.71\rho^4 + 16370.75\rho^5 - 48811.41\rho^6 + 68220.87\rho^7 - 46688\rho^8 + 12622.25\rho^9$$

$$B_* = 1.11 + 2.894\rho - 76.765\rho^2 + 1565.35\rho^3 - 11336.19\rho^4 + 44270.15\rho^5 - 97014.11\rho^6 + 118797.26\rho^7 - 76209.82\rho^8 + 19923.28\rho^9$$

$$C_* = 14.19 - 148.24\rho + 847.8\rho^2 - 2318.58\rho^3 + 3168.35\rho^4 - 1590.27\rho^5$$

The approximations $L^a$ and $L^a$ given by (17) and (18) hold within the entire flow region $0 \leq \rho \leq 1$, and only involve real elementary continuous functions (algebraic, exponential, logarithmic).

Fig.5 depicts the local flow components $L$ and $L^a$ defined by the integral representation (7) or the related approximation (17) for $0 \leq \rho \leq 1$ and six values of $0 \leq \alpha \leq 1$. Fig.6 similarly depicts the local flow components $L^*$ and $L^a$ given by the integral representation (10) or the related approximation (18) for $0 \leq \rho \leq 1$ and six values of $0 \leq \alpha \leq 1$.

Fig.7 depicts the local flow components $L_z$ and $L^a_z$ defined by the integral representation (7), (8a) or the related approximation (16), (17) for $0 \leq \rho \leq 1$ and six values of $0 \leq \alpha \leq 1$. Fig.8 similarly depicts the local flow components $L_h$ and $L^a_h$ given by the integral representation (8b), (10) or the related approximation (16), (18) for $0 \leq \rho \leq 1$ and six values of $0 \leq \alpha \leq 1$.

The functions $L$ and $L^a$ in Fig.5, the functions $L_*$ and $L_*^a$ in Fig.6, the functions $L_z$ and $L_z^a$ in Fig.7, and the functions $L_h$ and $L_h^a$ in Fig.8 cannot be distinguished.

5 ERRORS IN THE GREEN FUNCTION AND ITS GRADIENT

The errors associated with the approximations $L^a$ and $L^a$ given by (17) and (18) are now considered. The absolute errors $e$ and $e^*$ between the local flow components $L$ and $L_*$ and the corresponding approximations $L^a$ and $L^a$ in expressions (5) and (8) for the Green function and
Huiyu Wu, Yi Zhu, Chao Ma, Wei Li, Huiping Fu and Francis Noblesse

Figure 11: Approximate relative error $e' \equiv (L - L^a)/L_{v=0}$ between the local flow components $L$ and $L^a$ defined by the integral representation (7) or the approximation (17) for $0 \leq \rho \leq 1$ and $\alpha = 0$, 0.2, 0.4, 0.6, 0.8, 1.

Figure 12: Approximate relative error $e'_z \equiv (L_z - L^a_z)/L_{z=0}$ between the local flow components $L_z$ and $L^a_z$ defined by the integral representation (7), (8) or the approximation (16), (17) for $0 \leq \rho \leq 1$ and $\alpha = 0$, 0.2, 0.4, 0.6, 0.8, 1.

its gradient are defined as

$$4\pi e \equiv L - L^a \equiv L_z - L^a_z \quad 4\pi e^*_s \equiv L_s - L^a_s \equiv L_h - L^a_h$$

Along the vertical axis $\alpha = 1$, one has $L^a_s \equiv L_s$ and therefore $e^*_s \equiv 0$. The relative errors associated with the approximate local flow components $L^a$, $L^a_z$ and $L^a_h$ are defined as

$$e' \equiv \frac{L - L^a}{L} \equiv \frac{4\pi e}{L} \quad e'_z \equiv \frac{L_z - L^a_z}{L_z} \equiv \frac{4\pi e}{L_z} \quad e'_h \equiv \frac{L_h - L^a_h}{L_h} \equiv \frac{4\pi e^*_s}{L_h}$$

where $L^a_{v=0} \equiv L(h, v = 0)$, $L^a_{z=0} \equiv L_z(h, v = 0)$ and $L^a_{h=0} \equiv L_h(h, v = 0)$. The functions $e'$, $e'_z$ and $e'_h$ defined by (21) provide meaningful approximations to the relative errors associated with the approximations $L^a$, $L^a_z$ and $L^a_h$ within the entire flow region $0 \leq d$.

It can be shown that the absolute errors $e$ and $e^*_s$ defined by (19) behave as

$$e = O(d) \quad \text{as} \quad d \to 0$$

$$e = O(1/d^3) \quad \text{if} \quad \alpha \neq 0 \quad \text{or} \quad e = O(\log d/d^3) \quad \text{if} \quad \alpha = 0 \quad \text{as} \quad d \to \infty$$

$$e^*_s = O(d \log d) \quad \text{as} \quad d \to 0 \quad e^*_s = O(1/d^3) \quad \text{as} \quad d \to \infty$$

The relative errors (20) are now considered. The behaviors of the local flow components $L$ and $L_z$ defined by the integral representations (7) and (10) in the near-field and far-field limits
Figure 13: Approximate relative error $e_h' \equiv (L_h - L_h^a)/L_h^{a=0}$ between the local flow components $L_h$ and $L_h^a$ defined by the integral representation (8b), (10) or the approximation (16), (18) for $0 \leq \rho \leq 0.8$ and $\alpha = 0, 0.2, 0.4, 0.6, 0.8, 1$.

Figure 14: Approximate relative error $e_h' \equiv (L_h - L_h^a)/L_h^{a=0}$ between the local flow components $L_h$ and $L_h^a$ defined by the integral representation (8b), (10) or the approximation (16), (18) for $0 \leq \rho \leq 1$ and $\alpha = 0, 0.2, 0.4, 0.6, 0.8, 1$.

d $\rightarrow 0$ and $d \rightarrow \infty$ are considered in Noblesse [1]. In particular, one has

$$L_z \sim -\alpha/d^2 \text{ if } \alpha \neq 0 \text{ or } L_z \sim -2/d \text{ if } \alpha = 0 \text{ as } d \rightarrow 0$$

(24a)

$$L_z \sim \alpha/d^2 \text{ if } \alpha \neq 0 \text{ or } L_z \sim -4/d \text{ if } \alpha = 0 \text{ as } d \rightarrow \infty$$

(24b)

$$L_z \sim \alpha/d^2 \text{ if } \alpha \neq 0 \text{ or } L_z \sim -2/d \text{ if } \alpha = 0 \text{ as } d \rightarrow 0$$

(25a)

$$L_z \sim \alpha/d^2 \text{ if } \alpha \neq 0 \text{ or } L_z \sim -4/d \text{ if } \alpha = 0 \text{ as } d \rightarrow \infty$$

(25b)

Moreover, equations (8b) and (11b) yield $L_h = -4e^\alpha$ if $\alpha = 1$.

Expressions (22) and (24) show that the relative error $e^r$ defined by (20) behaves as

$$e/L = O(d^2) \text{ as } d \rightarrow 0$$

(27a)

$$e/L = O(1/d^2) \text{ if } \alpha \neq 0 \text{ or } e/L = O(\log d/d^2) \text{ if } \alpha = 0 \text{ as } d \rightarrow \infty$$

(27b)

One then has $e \ll L$ in both the near field and the far field. Expressions (22) and (25) similarly show that the relative error $e_h^\alpha$ defined by (20) behaves as

$$e/L_z = O(d^2) \text{ if } \alpha \neq 0 \text{ or } e/L_z = O(\log d/d^2) \text{ if } \alpha = 0 \text{ as } d \rightarrow 0$$

(28a)

$$e/L_z = O(1/d) \text{ if } \alpha \neq 0 \text{ or } e/L_z = O(\log d/d^2) \text{ if } \alpha = 0 \text{ as } d \rightarrow \infty$$

(28b)

One then has $e \ll L_z$ in both the near field and the far field. Expressions (23) and (26) likewise show that the relative error $e_h^\alpha$ defined by (20) behaves as

$$e_h/L_h = O(d^3 \log d) \text{ as } d \rightarrow 0 \text{ if } \alpha \neq 1 \text{ or } e_h/L_h = O(1/d) \text{ as } d \rightarrow \infty \text{ if } \alpha \neq 1$$

(29)

Along the vertical axis $\alpha = 1$, one has $e_h = 0$, as was already noted. One then has $e_h \ll L_h$ in both the near field and the far field.
Computations within the entire region $0 \leq d$ show that the absolute errors $e$ and $e_*$ defined by (19) vary within the ranges $-3 \times 10^{-4} < e < 3 \times 10^{-4}$ and $-2.4 \times 10^{-4} < e_* < 2.6 \times 10^{-4}$. Fig.9 and Fig.10 depict the errors $e$ and $e_*$ for $0 \leq \rho \leq 1$ and six values of $\alpha$. Fig.9 and Fig.10 show that the errors $e$ and $e_*$ vanish both as $\rho \to 0$ and as $\rho \to 1$, in accordance with (22) and (23). The errors $e$ and $e_*$ for the six values of $\alpha$ considered in Fig.9 and Fig.10 are of the same order of magnitude, and oscillate between positive and negative values that are distributed more or less evenly. Fig.10 shows that $e_* \equiv 0$ along the vertical axis $\alpha = 1$, as was already noted.

Fig.11 and Fig.12 depict the approximate relative errors $e'_e$ and $e'_e$ for $0 \leq \rho \leq 1$ and six values of $\alpha$. Computations within the entire region show that $e'_e$ and $e'_e$ vary within the ranges $-0.45 \times 10^{-2} < e'_e < 0.43 \times 10^{-2}$ and $-0.34 \times 10^{-2} < e'_e < 0.32 \times 10^{-2}$. Fig.13 and Fig.14 similarly depict the approximate relative error $e'_h$ for $0 \leq \rho \leq 0.8$ or $0 \leq \rho \leq 1$. The errors $e'_e$, $e'_e$ and $e'_h$ in Fig.11-Fig.13 are relatively small and of the same order of magnitude. Fig.14 shows that the error $e'_h$ is significantly larger in the far field $0.8 \leq \rho \leq 1$. Indeed, computations within the entire region show that $e'_h$ vary within the range $-4.33 \times 10^{-2} < e'_h < 4.73 \times 10^{-2}$. However, Fig.10 shows that the absolute error $e_*$ that corresponds to the relative error $e'_h$ is small in the range $0.8 \leq \rho \leq 1$ where $e'_h$ is large. Specifically, Fig.10 shows that the absolute error $|e_*|$ is smaller than $10^{-4}$ for $0.8 \leq \rho \leq 1$. Fig.11-Fig.14 show that the relative errors $e'_e$, $e'_e$ and $e'_h$ vanish in both the near field $\rho \to 0$ and the far field $\rho \to 1$. Moreover, Fig.13 and Fig.14 show that one has $e_* \equiv 0$ and therefore $e'_h \equiv 0$ along the vertical axis $\alpha = 1$.

6 CONCLUSIONS

The Green function $G$ in the classical theory of wave diffraction radiation by an offshore structure, or a ship at low speed, in deep water is expressed in the usual manner as the sum of the fundamental free-space singularity $-1/r$, a non-oscillatory local flow $L$, and waves $W$. The gradient of $G$ is similarly expressed as the sum of three basic components. The wave components $W$ and $W_h$ in these basic decompositions of $G$ and $\nabla G$ are expressed in terms of real functions of one variable, specifically the exponential function $e^v$, the Bessel functions $J_0(h)$ and $J_1(h)$ and the Struve functions $H_0(h)$ and $H_1(h)$. These functions are infinitely differentiable and can be readily evaluated; e.g. Hitchcock [16], Abramowitiz & Stegun [17], Luke [18], Newman [19].

The analytical approximations to the local flow components in the expressions for $G$ and $\nabla G$ given here are global approximations valid within the entire flow region, and only involve elementary continuous functions (algebraic, exponential, logarithmic) of real arguments. The analysis of the errors associated with the approximations to the local flow components given here and in Wu et al. [14] shows that the approximations are sufficiently accurate for practical purposes. These global approximations provide a particularly simple and highly efficient way of numerically evaluating $G$ and $\nabla G$ for diffraction radiation of regular waves in deep water.

REFERENCES


MODELLING OF ROTATING VERTICAL AXIS TURBINES USING A MULTIPHASE FINITE ELEMENT METHOD

Van Dang Nguyen*, Johan Jansson*†, Massimiliano Leoni†, Bärbel Janssen*, Anders Goude†, and Johan Hoffman*

*Department of Computational Science and Technology, School of Computer Science and Communication, KTH Royal Institute of Technology, Stockholm, Sweden.
e-mail: {vdnguyen, jjan, barbel, jhoffman}@kth.se

† Basque Center for Applied Mathematics, Bilbao, Spain.
e-mail: mleoni@bcamath.org.

‡ Uppsala University, Uppsala, Sweden.
e-mail: anders.goude@angstrom.uu.se

Key words: Vertical axis turbines, fluid-structure interaction, fluid-rigid body interaction, Unicorn solver, FEniCS-HPC, Navier-Stokes equations, multiphase finite element method.

Abstract. We combine the unified continuum fluid-structure interaction method with a multiphase flow model to simulate turbulent flow and fluid-structure interaction of rotating vertical axis turbines in offshore environments. This work is part of a project funded by the Swedish Energy Agency, which focuses on energy systems combining ecological sustainability, competitiveness and reliability of supply. The numerical methods used comprise the Galerkin least-squares finite element method, coupled with the arbitrary Lagrangian-Eulerian method, in order to compute weak solutions of the Navier-Stokes equations for high Reynolds numbers on moving meshes. Mesh smoothing methods help to improve the mesh quality when the mesh undergoes large deformations. The simulations have been performed using the Unicorn solver in the FEniCS-HPC framework, which runs on supercomputers with near optimal weak and strong scaling up to thousands of cores.

1 INTRODUCTION

Wind and water have been sources of energy for thousands of years, and one of the most utilized during the 17-th and 18-th centuries [10]. Nowadays, the increased interest in renewable energy resources has caused an increase in wind and water power utilization, and improvement of wind and hydraulic turbine technology. The function of a wind turbine is the result of a complex interaction of its components and subsystems such as the blades, rotor, tower, foundation, power train, control system, etc. [1]. Vertical axis turbines have long been used to convert kinetic energy from wind/water into electrical power, and numerical solution of the Navier-Stokes equations for simulation of wind turbines may explain some physical phenomena and
Modelling of rotating vertical axis turbines

give some guidance for improvement. Simulation of a rotating turbine is a challenging problem, considered an open problem in all its complexity, even though some impressive results have been achieved using stabilized finite element methods [2, 3].

In this paper, we combine the unified continuum fluid-structure interaction method [7] with a multiphase flow model [9] to simulate turbulent flow and fluid-structure interaction of rotating turbines in offshore environments. This work is part of a project funded by the Swedish Energy Agency, which focuses on energy systems combining ecological sustainability, competitiveness and reliability of supply. The numerical method used comprises the Galerkin least-squares finite element method, coupled with the arbitrary Lagrangian-Eulerian method in order to allow us to compute weak solutions of the Navier-Stokes equations for high Reynolds numbers on moving meshes. Mesh smoothing methods help to improve the mesh quality when the mesh undergoes large deformations. The simulations have been performed using the Unicorn solver in the FEniCS-HPC framework, which runs on supercomputers with near optimal weak and strong scaling up to thousands of cores.

The paper is organized as follows. We first introduce in Section 2 the vertical axis turbine used for all simulations in the paper. In Section 3 we consider a simplified model based on fluid-rigid body interaction in which the turbine rotates with a given rotational speed applied to the computation mesh. In Section 4 we study a full fluid-structure interaction model. The computer implementation is discussed in Section 5. In Section 6 we present some preliminary results and suggest future directions.

2 A VERTICAL AXIS TURBINE

The turbine under consideration is a single 3-bladed H-rotor turbine, with a radius of 3.24 m and a blade length of 5 m (Fig. 1a, is reproduced from [5]). The blades are pitched outwards with a chord length of 0.25 m at the middle of the blade. For simplification, we assume that the turbine axis is coincident with the $z$-axis and that the turbine $\Omega^T$ is placed in a cylinder $\Omega^C$ (Fig. 1b)

$$\Omega^C = \{(x, y, z) \in \mathbb{R}^3 \mid x^2 + y^2 \leq R^2, z \in [-L/2; L/2]\}. \quad (1)$$

Here we choose $R = 30$ m and $L = 20$ m, with the turbine axis placed in the center-line of the cylinder domain. Let $f$ and $s$ indicate fluid and structure parts respectively and for fluid-structure interaction we denote $\Omega^f = \Omega^C \setminus \Omega^T$ and $\Omega^s = \Omega^T$.

3 FLUID-RIGID BODY INTERACTION MODEL

We start with a simplified model where the structure is assumed to be a rigid body, so that we remove $\Omega^s$ from the domain and solve only for the fluid domain $\Omega = \Omega^f$ where the turbine is forced to rotate with a given rotational velocity. The incompressible Navier-Stokes equations Eq. 2 need to be solved in the fluid part, $\Omega = \Omega^f$.

$$\begin{align*}
\dot{u} + (u \cdot \nabla)u - \nu \Delta u + \nabla p &= f & \text{in } \Omega \times I \\
\nabla \cdot u &= 0 & \text{in } \Omega \times I \\
u(r, 0) &= u_0 & \text{in } \Omega
\end{align*} \quad (2)$$
Figure 1: A vertical axis turbine reproduced from [5] (a), placed in a cylinder (b).

Here, $\mathbf{u}$ is the unknown velocity, $p$ is the unknown pressure, $\nu$ is the kinematic viscosity computed as the ratio between the dynamic viscosity $\mu$ and the density $\rho$, i.e. $\nu = \mu/\rho$, and $\mathbf{f}$ is a given body force. Eq. (2) is subjected to the slip velocity boundary condition

$$\mathbf{u} \cdot \mathbf{n} = 0 \tag{3}$$

or the no-slip velocity boundary condition

$$\mathbf{u} = 0 \tag{4}$$

on boundary of the fluid domain at the turbine surface $\Gamma^T$. Here $\mathbf{n}$ is the outward unit normal of the fluid boundary and $\sigma$ denotes the stress tensor.

The inflow velocity boundary condition $\mathbf{u} = (u_i, 0, 0)$ and the outflow condition $p = 0$ are imposed on $\Gamma^i$ and $\Gamma^o$ respectively. These boundaries are defined as follows

$$\Gamma^T = \partial \Omega \cap \Omega^T \tag{5}$$
$$\Gamma^i = \{ x \in \partial \Omega \mid x \leq 0 \} \setminus \Gamma^T \tag{6}$$
$$\Gamma^o = \{ x \in \partial \Omega \mid x > 0 \} \setminus \Gamma^T \tag{7}$$

To solve this problem, the simplest algorithm we can think of is to manually rotate the computational domain. At each time step, we update the mesh and the corresponding finite element
function spaces. However, the solution from the previous time step needs to be interpolated to the new function spaces. This process is expensive and accumulates interpolation errors. To overcome the problem, the arbitrary Lagrangian-Eulerian (ALE) method \[4\] is applied. This method automatically projects the solution from the previous time step to the current function space without using interpolation. The Navier-Stokes’ momentum equation \[2\] formulated in ALE coordinates with a mesh velocity \(\beta\) reads

\[
\dot{u} + (u - \beta) \cdot \nabla u - \nu \Delta u + \nabla p = f
\]  

Galerkin least-squares space-time FEM \[6\] is used to obtain an efficient method for the discretization of a rotating turbine problems. Let \(0 = t_0 < t_1 < \cdots < t_N = T\) be a time partition associated with the time intervals \(I_n = (t^{n-1}, t^n)\) of length \(k_n = t^n - t^{n-1}\). Let \(Q \subset H^1(\Omega), Q_0 \subset H^1_0(\Omega)\) and \(V = [Q_0]^d\). The ALE finite element method with least-squares stabilization is stated as the following. For all time intervals \(I_{n+1}\), find \((U^{n+1}, P^{n+1}) \in V_h \times Q_h\) such that

\[
\left( (U^{n+1}_h - U^n_h) k_{n+1}^{-1} + (\hat{U}^n_h - \beta) \cdot \nabla \hat{U}^n_h, v \right) + \left( \nu \nabla \hat{U}^n_h, \nabla v \right) - \left( P^{n+1}, \nabla \cdot v \right) + \left( \nabla \cdot \hat{U}^n_h, q \right) + SD_\delta \left( \hat{U}^n_h, P^{n+1}; v, q \right) = (f, v) \]  

where

\[
SD_\delta \left( \hat{U}^n_h, P^{n+1}; v, q \right) = \left( \delta_1 \left( (\hat{U}^n_h \cdot \nabla) \hat{U}^n_h + \nabla P^{n+1}_h - f \right), (\hat{U}^n \cdot \nabla) v + \nabla q \right) + \left( \delta_2 \nabla \cdot \hat{U}^n_h, \nabla \cdot v \right) \]  

for all \((v, q) \in V_h \times Q_h\). Here \(\delta_1\) and \(\delta_2\) are stabilization parameters given by positive constants \(C_1, C_2\), the time step size \(k_n\) and the mesh size \(h_n\)

\[
\delta_1 = C_1 \left( \frac{1}{k_{n+1}^2} + \frac{|U^{n-1}_h|^2}{h_n^2} \right)^{-1/2}
\]

\[
\delta_2 = C_2 h_n |U^{n-1}| \]  

Since the inflow and outflow boundary conditions are not moving with the mesh, they need to be updated for each new time step.

4 FLUID-STRUCTURE INTERACTION MODEL

Including structure deformation, the rotating turbine can be described using a unified continuum fluid-structure interaction model \[7\]:

\[
\begin{cases}
\rho \left( \dot{u} + (u \cdot \nabla) u \right) + \nabla \cdot \sigma = f & \text{in } \Omega \times I \\
\nabla \cdot u = 0 & \text{in } \Omega \times I \\
\theta + (u \cdot \nabla) \theta = 0 & \text{in } \Omega \times I \\
u(\cdot, 0) = u_0 & \text{in } \Omega
\end{cases}
\]  

953
The computational domain is $\Omega = \Omega_f \cup \Omega_s$, $\Gamma^{fs} = \overline{\Omega_f} \cap \overline{\Omega_s}$, the phase function $\theta$ defines the structure and fluid domains

$$\theta(x, t) = \begin{cases} 
1 & \text{if } x \in \Omega_f \\
0 & \text{if } x \in \Omega_s 
\end{cases}$$

(13)

For a Newtonian fluid and an incompressible Neo-Hookean solid, the stress $\sigma$ is computed as the following

$$\sigma = -\sigma_D + p I$$

$$\sigma_D = \theta \sigma_f + (1 - \theta) \sigma_s$$

$$\sigma_f = 2 \mu_f \varepsilon(u)$$

$$\sigma_s = 2 \mu_s \varepsilon(u) + \nabla u \sigma_s + \sigma_s \nabla u^T$$

(14)

The no-slip velocity boundary condition in this setting means that

$$u_f = u_s$$

(15)

on $\Gamma^{fs}$ and the slip condition is

$$u_f \cdot n = u_s \cdot n$$

(16)

We also need to assure the force balance condition at the interface, i.e $\sigma_f \cdot n = \sigma_s \cdot n$.

The variational form of the FSI model in ALE coordinates reads

$$\left( \rho \dot{u}, v \right) + \left( (u - \beta) \cdot \nabla u, v \right) - \left( \rho, \nabla \cdot v \right) + \left( \alpha \varepsilon(u), \varepsilon(v) \right) + \left( \delta_1 (u - \beta) \cdot \nabla u, (u - \beta) \cdot \nabla v \right)$$

$$+ \left( \delta_2 \nabla p, \nabla q \right) - \left( \nabla \cdot u, q \right) = \left( f, v \right) - (1 - \theta) \left( \sigma_D^{n-1}, \nabla v \right)$$

(17)

where $v, q$ are test functions and

$$\rho = \theta \rho_f + (1 - \theta) \rho_s$$

$$\alpha = \theta 2 \nu + (1 - \theta) kE$$

Since the solid part of the mesh moves, the mesh at the interface between the solid and the fluid becomes ill conditioned. Laplacian mesh smoothing techniques have been used to redistribute the vertices of the mesh in the fluid domain. The technique works by moving all the internal vertices to the center of all its neighbors.

5 IMPLEMENTATIONS

The two numerical models described above have been implemented in Unicorn and the high performance branch of the finite element problem solving environment DOLFIN-HPC which is optimized for distributed memory architectures using a hybrid MPI+OpenMP approach with efficient parallel I/O. Both Unicorn and DOLFIN-HPC are written in C++ and the implementation has proven to be portable across several different architectures, such as ordinary Unix/Linux workstations. We use ParMETIS for mesh partitioning.

The first model has been implemented with both the slip condition (Eq. 3) and the no-slip condition (Eq. 4), whereas the second model has only been implemented with the no-slip condition in Eq. 16.
6 PRELIMINARY RESULTS AND DISCUSSION

The proposed methods were implemented in the Unicorn solver for parallel adaptive finite element simulation of turbulent flow and fluid-structure interaction. The solver showed near optimal weak and strong scaling results when simulating turbulent flow on massively parallel machines [8]. The adaptive mesh refinement in the solver, however, is disabled for this case since the $h-$adaptivity does not work properly for rotating computational domains. The use of $r-$adaptivity may be a better choice for the ALE case of rotating domain and we leave it for future works. The simulations were carried out on the Beskow supercomputer at the KTH Royal Institute of Technology, that is a Cray XC40 system, based on Intel Haswell processors and Cray Aries interconnect network. Each Beskow node has 32 cores divided into two sockets, with 16 cores each, and thus the total number of cores is $53632$. The available RAM for each node is 64 GB DDR3 SDRAM.

The volume meshes were generated with ANSA from Beta-CAE Systems with 556789 vertices for the fluid part and 116622 vertices for the structure part. We note that in the fluid-rigid body interaction model, only the fluid part of the mesh is used whereas in the fluid-structure interaction model, the computational mesh unifies the fluid mesh and the structure mesh.

Numerical simulations show that the fluid-rigid body interaction model works efficiently. We show in Fig. 2 drag forces for three rotational velocities: fixed turbine, $\pi/200$ rad/s and $\pi/20$ rad/s. A larger rotational velocity gives larger oscillations. Some snapshots in time for the two rotational velocities are visualized in Fig. 3 and Fig. 4 respectively.

![Figure 2: Drag forces for three rotational velocities: fixed turbine, $\pi/200$ rad/s and $\pi/20$ rad/s.](image-url)
Figure 3: Rotating turbine at rotational speed $\pi/200$ rad/s, for 4 snapshots in time: horizontal cuts (left) and vertical cuts (right).

Fig. 5a shows a vertical cut of the computed velocity for the fluid-structure interaction model where the no-slip velocity Eq. 15 is applied. Under the effect of the inflow velocity, the turbine rotates for a while (Fig. 5b) before stopping in the position shown in Fig. 5c. One of the reasons may be because the no-slip boundary condition (15) is not suitable. A Nitsche method for the slip condition (16) is under consideration and it is expected to solve the problem.
Figure 4: Rotating turbine at rotational speed $\pi/20 \text{rad/s}$, for 4 snapshots in time: horizontal cuts (left) and vertical cuts (right).

ACKNOWLEDGEMENTS

This research has been supported by the European Research Council, the Swedish Energy Agency, the Basque Excellence Research Center (BERC 2014-2017) program by the Basque Government, the Spanish Ministry of Economy and Competitiveness MINECO: BCAM Severo
Modelling of rotating vertical axis turbines

Figure 5: A vertical cut of the computed velocity (a). Under the effect of the inflow velocity, the turbine only rotates for a while (b) before stopping in the position shown in (c).

Ochoa accreditation SEV-2013-0323, the ICERMAR ELKARTEK project of the Basque Government, the Projects of the Spanish Ministry of Economy and Competitiveness with reference MTM2013-40824-P and MTM2016-76016-R. We acknowledge the Swedish National Infrastructure for Computing (SNIC) at PDC – Center for High-Performance Computing for awarding us access to the supercomputer resource Beskow. The initial volume mesh was generated with ANSA from Beta-CAE Systems S. A., who generously provided an academic license for this project.

REFERENCES


A NUMERICAL STUDY OF ADDED RESISTANCE, SPEED LOSS AND ADDED POWER OF A SURFACE SHIP IN REGULAR HEAD WAVES USING COUPLED URANS AND RIGID-BODY MOTION EQUATIONS

MARINE 2017

SHAWN ARAM* AND SUNG-EUN KIM**

Naval Surface Warfare Center, Carderock Division
West Bethesda, MD, USA
e-mail: shawn.aram@navy.mil*; sungeun.kim@navy.mil**

Key words: URANS, 6-DOF ship motion, added resistance, added power, head waves

Abstract. Coupled solutions of two-phase Unsteady Reynolds-Averaged Navier-Stokes equations (URANS) and six degrees-of-freedom (6-DOF) rigid-body motion (RBM) equations are pursued to compute ship’s speed loss and added power for a self-propelling ship in regular head sea. A case study is presented for a 1/49 scale model of the ONR Tumblehome (ONRT) (Model 5613) which was tested in the towing tank of the Iowa Institute of Hydraulic Research (IIHR). The computations were carried out for various combinations of towed and self-propelled conditions, and in calm water and regular head waves with a range of wave length and amplitude. The case study demonstrates that the coupled URANS and RBM solution approach can predict added resistance, speed loss and added power of a ship cruising in head waves with commendable accuracy and shed light on the complex interactions among the ship, propeller, waves and flow-fields.

1 INTRODUCTION

Ship’s resistance and engine power to sustain ship’s speed in seaways are augmented due to complex non-linear interactions between the ship and the ambient sea (waves). Ship designers, in early design stage, use an ad hoc “sea margin” to account for the effects of seaways in selecting propeller and engine. A numerical tool capable of accurately predicting added resistance and power of a ship cruising in waves would greatly help select its powering (margin) requirement and determine the optimal operating point that can maximize the energy efficiency.

For seakeeping analysis, strip theory-based methods have long been used. More recently, nonlinear time-domain three-dimensional (3D) panel methods have started being used widely [1]. A more physics-based avenue to seakeeping analysis is offered by coupled solutions of two-phase unsteady Reynolds-Averaged Navier-Stokes equations and six degrees-of-freedom rigid-body motion (RBM) equations [2, 3, 4]. The URANS approach also avails itself of including the effects of propulsors, either explicitly or approximately. By accounting for all the nonlinear effects in hydrodynamic forces and moments and the resulting ship motions, and the effects of fluid viscosity and turbulence, the coupled URANS-RBM method is believed not only able to predict added resistance and speed loss more accurately but also to provide valuable insights into the physical mechanisms underlying added resistance and power.

This material is declared a work of the U.S. Government and is not subject to copyright protection in the United States. Approved for public release; distribution is unlimited.
The objectives of this study are:
- To validate a coupled URANS-RBM solver developed for high-fidelity prediction of added resistance, speed loss and added power on ships cruising in regular head sea.
- To conduct a detailed analysis of the interactions among ship hull, propeller and waves for a 1/49 scaled model of the ONR Tumblehome (ONRT) (Model 5613) in order to shed light on the physical mechanisms leading to added resistance, speed loss and added power.

2 SHIP MODEL

The ONR Tumblehome (ONRT) model 5613, shown in Figure 1, is considered for the present study. It is a fully appended 1/49 scale model equipped with a skeg, bilge keels, twin rudders, shafts and two 4-bladed propellers mounted with shaft brackets. Each propeller has 4 fixed-pitch type blades with inward direction of rotation (view from bow/stern). The main particulars of the ship are given in Table 1.

![Figure 1: ONR Tumblehome (ONRT) model 5613](image)

Table 1: Particulars of ONRT model scale hull

<table>
<thead>
<tr>
<th>Main Particulars</th>
<th>Model Scale</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacement, ( \Delta ) (kg)</td>
<td>72.6</td>
</tr>
<tr>
<td>Waterline Length, ( L ) (m)</td>
<td>3.147</td>
</tr>
<tr>
<td>Waterline Beam, ( B_{WL} ) (m)</td>
<td>0.384</td>
</tr>
<tr>
<td>Draft, ( T ) (m)</td>
<td>0.112</td>
</tr>
<tr>
<td>Wetted Surface Area, ( S ) (m(^2))</td>
<td>1.5</td>
</tr>
<tr>
<td>( L_{CB} ) (m aft of FP)</td>
<td>1.625</td>
</tr>
<tr>
<td>( V_{CG} ) (m from keel)</td>
<td>0.156</td>
</tr>
<tr>
<td>Yaw Inertia (( K_{yy}/L ))</td>
<td>0.246</td>
</tr>
<tr>
<td>Propeller Diameter, ( D_p ) (m)</td>
<td>0.1066</td>
</tr>
<tr>
<td>Propeller Shaft Angle (deg)</td>
<td>5</td>
</tr>
</tbody>
</table>

Free-running tests of the hull at various maneuvering conditions in calm water and regular waves were performed at the IIHR Wave Basin Facility [5, 6]. The experimental data are used to validate the predictions in this paper.

3 COMPUTATIONAL METHOD

3.1 CFD Solver – NavyFOAM

All the URANS-based simulations were conducted using an in-house CFD framework, NavyFOAM. NavyFOAM is a suite of CFD codes developed using OpenFOAM®, an open-
source continuum mechanics software library written in C++. OpenFOAM makes use of object-oriented programming techniques offered by the C++ language that allow one to maximize code re-use, adopt layered development, expedite building top-level applications, and make runtime-selection of numerical schemes, solution algorithms, physical models, and file I/O. NavyFOAM offers additional libraries in areas such as discretization schemes and physical models. Several top-level solvers for single- and multi-phase flows have also been added in NavyFOAM aiming at marine applications including underwater vehicles, surface ships, and propulsors [7].

The Navier-Stokes solvers in NavyFOAM employ a cell-centered finite-volume method based on a multi-dimensional linear reconstruction scheme that permits use of arbitrary polyhedral cells. The free surface (air-water interface) is resolved by a two-phase, single-fluid Volume-Of-Fluid (VOF) method [8]. Among the schemes offered in NavyFOAM to discretize the advection term in the volume-fraction equation, the modified high resolution interface capturing (MHRIC) scheme [8] is employed in the current study. The advection term in the momentum equation is discretized using the 2nd-order upwind scheme with skewness correction employed for the diffusion term.

The continuity, momentum, volume-of-fluid and turbulence equations are solved implicitly in a segregated manner. The 2nd-order backward-differencing scheme is used for temporal discretization. The SST k-ω model is employed to model the turbulence.

The “rigid-body mesh” option in NavyFOAM is used to model 3-DOF (straight ahead self-propulsion, free to surge, heave and pitch) motions of the ONRT model. The coupled solutions of the URANS and 6-DOF motion equations are obtained using a predictor-corrector-based, iterative algorithm that not only preserves a formal 2nd-order temporal accuracy, but ensures stability of the coupled solutions. The tight coupling in the solution algorithm is achieved by nested loops where each coupling loop contains an outer iterative loop for the flow equations. Three coupling loops and two outer iterations are used in this study.

The effects of propellers could be modeled in NavyFOAM using one of the following techniques:

- Direct method based on the Generalized Grid Interface (GGI) technique, in which the actual propeller rotation is modeled. The GGI, which is an efficient and scalable algorithm for sliding grids, allows for a time-accurate solution to be obtained for propeller-hull interactions. This method uses a special algorithm to compute weights for solution interpolation between two overlapping non-conformal surfaces. It should be pointed out that this method is computationally expensive due to the very small time-step size necessary to resolve the time scales associated with propeller-hull interactions.

- Body-force model, where the propeller geometry is excluded from the model. Given the orientation and location of the propeller, the surrogate model dynamically updates the thrust and torque using the characteristics (K_T, K_Q vs. J) and the ship speed or local inflow velocity evaluated at each time instant. Variation in thrust and torque with the average inflow rate and impeller RPM (K_T and K_Q vs. J) are derived from available propeller performance data. Using the body-force model, a volumetric body-force which represents the effect of actual rotating propeller is distributed at the location of propeller. The Hoekstra’s actuator disk model [9] is adopted in NavyFOAM to model the radial distribution of body force.
Both the direct method and the body-force model were used in this study to model the thrust generated by propellers and to evaluate the accuracy and efficiency of each method.

A wave-making library modified from the Waves2Foam package and integrated into NavyFOAM, was used to generate a single 1st-, 2nd- and 5th-order Stokes waves. It also supports a wave relaxation zone, within which the analytical flow and wave fields (flow velocity and volume-fraction) are implicitly blended with the solutions to the transport equation, using a predefined blending function (exponential function in this study). The computational time can be saved by using a wave-making zone along with the wave boundary condition specified at the inlet of the computational domain. The wave damping zone can be used near the outlet of the domain to minimize the wave reflections from the exit boundary. This is achieved by adding an artificial damping body-force term to the momentum equation. As a result, the flow velocity at the end of the damping zone tends to vanish, and the free surface recovers its calm-water position.

3.2 Computational Domain and Grids

The computational domain extends 1.5L forward of the forward perpendicular, 3.0L aft of the aft perpendicular, 2.0L to the side, L from top to waterline and 1.5L from waterline to bottom, where L is the waterline length.

HEXPRESSTM, a commercial meshing software package from NUMECA, was used to generate non-conformal body-fitted full hexahedral unstructured meshes. Quadrilateral elements were predominantly used to construct the hull surface in combination with the local refinements to properly capture the sharp edges (see Figure 2). The largest cell size (edge length) of the background grid is 0.75m (~L/4) in all three directions. Figure 3 shows the grid distribution on the centerline plane. The grid nodes are uniformly distributed along streamwise (x) and vertical (z) directions within the wave relaxation zone which extends 1.0L from the inlet boundary. Extensive grid sensitivity studies with a 2D wave tank and the Korean Container Ship (KCS) hull in regular waves with various wave slopes were conducted to determine a grid resolution required to accurately resolve the waves. The grid nodes are stretched along the x-direction inside the damping zone in order to damp the wave reflections at the outlet boundary. The “rigid-body dynamic mesh” method, in which the entire grid moves (translates and rotates) with the body, requires a band of the free-surface refinement zone to be sufficiently large not to lose the grid resolution near the upstream inlet boundary. The choice of the band-width depends on the incoming wave height as well. The near-wall resolutions are commensurate with the wall functions. On the hull surface, y' ranges from 50 – 70. The total number of cells for the case of body-force model varies between 5.2 million and 6.34 million elements, depending on the wave slope.

Figure 4 shows the surface grid on the propeller blades and the volume grid in the cylindrical rotating zone surrounding the propeller. As shown, the grid properly resolves the blades tips and edges. The total number of cells in the rotating zone is 1.5 million. The total cell count for the entire computational domain including the propeller and hull grids is 7.3 million.
The sensitivity of the computational results to the grid resolution was also examined by doubling the grid resolution in all three directions at the critical zones including near hull surface, propeller (body-force model), air-water interface, and wake regions. The total number of elements of this refined grid is 24.5 million.

3.3 Simulation Conditions

Table 2 summarizes the simulation conditions considered in this study. In all our computations, the ship was set free to surge, heave and pitch (3-DOF), but was constrained in sway and yaw. It should be noted that, in the model test, the model ship was set free to yaw, and the rudder angle was constantly adjusted using a controller to keep the model in a straight course. The self-propulsion simulations were performed first in calm-water to provide a reference condition for the subsequent added-power and added-resistance studies (Cases 1 and 2). The computational results (ship’s speed, sinkage and trim) were compared against the experimental data obtained at the IIHR’s Wave Basin Facility [5, 6]. Both the body-force model and the direct method were used to simulate the self-propulsion conditions at a constant propeller rotational speed of 538 RPM (Case 1). The simulation with a variable
RPM for the target ship speed of 1.11 m/s (Re = 3.4 \times 10^6, Fr = 0.2) was also performed using the body-force model (Case 2) to be used for the added power analysis. The resistance prediction of the ship towed in calm water was conducted as well for use in the propeller-hull interaction study (Case 3).

To represent regular head seas, three different wave-lengths of $\lambda/L=0.5$, 1.0 and 2.0 at a constant wave-slope of $h/L=0.02$ were considered. Thus, the corresponding wave-height also increases with the wave-length. Cases 4 to 6 were to study the ship’s speed loss in head waves at a fixed propeller speed (538 RPM). Except for Case 5 ($\lambda/L=1.0$), the body-force model was exclusively used to simulate the propeller effects. The speed-loss predictions for Case 5 ($\lambda/L=1.0$) were validated against the experimental data of Sanada et al. [5, 6] using both the body-force model and direct method for propeller.

For the added-power study, a Proportional Integral Derivative (PID) controller was used to maintain the target ship speed of 1.11 m/s as reported in the IIHR’s experiments for the calm water condition (Case 1), by varying the propeller RPM. Similar to the speed-loss study, the added power was computed for three wave lengths of $\lambda/L=0.5$, 1.0 and 2.0 (Case 7-9). Resistance predictions of the hull at these wave conditions (case 10-12) were also performed to determine the added resistance due to waves and to aid the propeller-hull interaction analyses for both constant and variable propeller RPM conditions. Each of Cases 10 to 12 ran with two different ship speeds: (1) the target speed of 1.11m/s (for the added resistance and added power study), (2) the ship speed determined by the self-propulsion simulation at constant propeller RPM of 538 (for the speed loss study).

<table>
<thead>
<tr>
<th>Case no.</th>
<th>Wave condition</th>
<th>Motion condition</th>
<th>RPM</th>
<th>Propeller action</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Calm water</td>
<td>Self-prop</td>
<td>538</td>
<td>on* (body-force &amp; direct)</td>
</tr>
<tr>
<td>2</td>
<td>Calm water</td>
<td>Self-prop</td>
<td>Var</td>
<td>On</td>
</tr>
<tr>
<td>3</td>
<td>Calm water</td>
<td>Tow</td>
<td></td>
<td>Off</td>
</tr>
<tr>
<td>4</td>
<td>Reg. wave $\lambda L=0.5$</td>
<td>Self-prop</td>
<td>538</td>
<td>On</td>
</tr>
<tr>
<td>5</td>
<td>Reg. wave $\lambda L=1.0$</td>
<td>Self-prop</td>
<td>538</td>
<td>on* (body-force &amp; direct)</td>
</tr>
<tr>
<td>6</td>
<td>Reg. wave $\lambda L=2.0$</td>
<td>Self-prop</td>
<td>538</td>
<td>On</td>
</tr>
<tr>
<td>7</td>
<td>Reg. wave $\lambda L=0.5$</td>
<td>Self-prop</td>
<td>Var</td>
<td>On</td>
</tr>
<tr>
<td>8</td>
<td>Reg. wave $\lambda L=1.0$</td>
<td>Self-prop</td>
<td>Var</td>
<td>On</td>
</tr>
<tr>
<td>9</td>
<td>Reg. wave $\lambda L=2.0$</td>
<td>Self-prop</td>
<td>Var</td>
<td>On</td>
</tr>
<tr>
<td>10</td>
<td>Reg. wave $\lambda L=0.5$</td>
<td>Tow (2 speeds)</td>
<td>-</td>
<td>off</td>
</tr>
<tr>
<td>11</td>
<td>Reg. wave $\lambda L=1.0$</td>
<td>Tow (2 speeds)</td>
<td>-</td>
<td>off</td>
</tr>
<tr>
<td>12</td>
<td>Reg. wave $\lambda L=2.0$</td>
<td>Tow (2 speeds)</td>
<td>-</td>
<td>off</td>
</tr>
</tbody>
</table>

* denote cases where the propeller was modeled using both the direct method and the body-force approach.
4 RESULTS AND DISCUSSION

4.1 Calm Water Condition

The predicted ship speed, sinkage and trim in calm water (Case 1) at the constant propeller RPM are compared with the experimental data in Table 3. The prediction of the ship speed using the body-force model is in excellent agreement with the measurement (0.5% difference), while the direct method under-predicted it by 5.0%. This may seem somewhat puzzling, inasmuch as one would expect better prediction from the direct method. At this point, we can only surmise that the experimental uncertainties arising from the use of a fairly small model (model ship length of 3.15 m and propeller diameter of 0.1 m), the effects of yaw allowed in the experiment, and modeling error could potentially be attributable to the discrepancy. Thus, the seemingly excellent prediction by the body-force model may well be fortuitous. The sinkage and trim predictions seem reasonable, although the percentage differences from the measurements are quite large due to the very small values of the trim and sinkage.

<table>
<thead>
<tr>
<th>Wave condition</th>
<th>ΔC Dü (×10³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>λ/L=0.5</td>
<td>0.375</td>
</tr>
<tr>
<td>λ/L=1.0</td>
<td>4.363</td>
</tr>
<tr>
<td>λ/L=2.0</td>
<td>1.884</td>
</tr>
</tbody>
</table>

Another way of validating the prediction for the IIHR’s experiment in calm-water condition is using the PID controller to adjust the propeller RPM to maintain the target ship speed of 1.11 m/s (Case 2). The present computation with the body-force model showed that the required RPM is found to be 541, which is slightly higher (0.5%) than the RPM used in the experiment.

4.2 Regular Wave Conditions

4.2.1 Added Resistance

Before presenting the speed loss and added power, the added resistance due to the head waves was computed using the results of the computations carried out for the towed conditions without the propellers (Cases 3, 10, 11, 12). Table 4 summarizes the results.

Table 4: Added resistance of ONRT due to waves towed at a constant speed of 1.11 m/s (Fr = 0.2)

4.2.2 Speed Loss

The computations were carried out with three different wave lengths (and heights) to predict the speed loss. Figure 5 shows the time histories of the ship speed, sinkage and trim predicted for the case of λ/L=1.0 and h/L=0.02 (Case 5) with the propeller operating at a fixed (538) RPM, along with the experimental data. Note that the ship speed plotted here was made dimensionless using the ship speed in calm water (denoted “V” in the plot). The wave encounter period, T_e, is used to non-dimensionalize time throughout this paper. The time-averaged ship speed is seen to be slightly under-predicted closely by both the body-force
method (1.7%) and the direct method (2.4%), respectively. However, the oscillation amplitude of the measured ship speed is almost twice as large as the predictions. The cause for the much larger oscillation amplitude of the measured ship speed is not clear, other than the plausible effects of the yaw motion of the ship model. Interestingly, the oscillation amplitude of the ship speed is also under-predicted by all others who tacked this case [5, 6, 10, 11]. The predicted amplitudes of the ship motion (heave and pitch) are in close agreement with the measurements, being slightly under-predicted.

The grid sensitivity of the predictions was also studied using the body-force model. Figure 6 shows the time histories of the ship speed, sinkage and trim obtained using 7.3 million cell (coarse grid) and 24.5 million cell (fine grid) grids. The predictions with both grids almost fall on top of each other, which indicate that the coarse mesh practically gives grid-convergent solutions.

Figure 5: Comparison of time histories ship speed, sinkage and trim in regular waves at $\lambda/L=1.0$ and $h/L=0.02$ conditions between CFD and EFD at fixed propeller RPM of 538

Figure 6: Grid sensitivity study for self-propulsion simulations in regular waves at $\lambda/L=1.0$, $h/L=0.02$ conditions and fixed propeller RPM of 538

Figure 7: Iso-surface of $Q$ (the second invariant of velocity deformation tensor) colored by the flow velocity magnitude at the stern region at minimum (left figure) and maximum surge speed (right figure)
The iso-surfaces of the Q colored by the velocity magnitude are shown in Figure 7 near the ship’s stern at two time instants corresponding to the moments when the ship attains the minimum and the maximum surge speed, respectively. The main flow features expected at the ship’s stern, such as the hub vortices and blade-tip vortices from the propeller are captured. It is also seen that the velocity magnitude along the tip vortices is noticeably greater at the maximum surge velocity than at the minimum surge velocity.

We summarized in Table 5 the hull axial force and its breakdowns to the pressure and frictional components for the calm-water and the wave conditions. Note that the hull forces were non-dimensionalized using the time-averaged ship speeds obtained for individual wave conditions. Also note that the hull force was computed by integrating the pressure and shear stresses over the hull with the propeller operating at a constant RPM. Thus, the hull axial force reported here accounts for the effects of the propeller. As can be seen, it is the pressure force component that is more affected by the change in the wave length and height. Both the total hull force along with its pressure and viscous contributions are the greatest at $\lambda/L=1.0$ (“resonance condition”), among all the flow conditions studied here. Figure 8 shows the time histories of the non-dimensional pressure and viscous components of the hull axial force at the calm water and wave conditions. The oscillation amplitudes of the pressure force coefficient increase with the wave length and height. The oscillation amplitudes of the viscous force coefficient at $\lambda/L=1.0$ and 2.0 are substantially larger than that for $\lambda/L=0.5$ presumably due to the changes in the wetted surface area. The oscillation amplitude of the pressure force coefficient is much larger than the viscous force coefficient at all wave conditions.

Table 5: Time average of non-dimensional pressure and viscous components and total of hull axial force coefficients ($\times 10^3$) at calm water and wave conditions and a fixed propeller RPM of 538

<table>
<thead>
<tr>
<th>Wave condition</th>
<th>$\bar{C}<em>{H</em>{\text{NetP}}}$</th>
<th>$\bar{C}<em>{H</em>{\text{Netvis}}}$</th>
<th>$\bar{C}_{\text{Tot}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calm water</td>
<td>0.813</td>
<td>3.6</td>
<td>4.413</td>
</tr>
<tr>
<td>$\lambda/L=0.5$</td>
<td>1.340</td>
<td>3.996</td>
<td>5.336</td>
</tr>
<tr>
<td>$\lambda/L=1.0$</td>
<td>5.745</td>
<td>4.804</td>
<td>10.549</td>
</tr>
<tr>
<td>$\lambda/L=2.0$</td>
<td>3.23</td>
<td>4.347</td>
<td>7.577</td>
</tr>
</tbody>
</table>

Figure 8: Time histories of non-dimensional pressure and viscous components of the axial hull force in head waves at $\lambda/L=0.5$, 1.0, 2.0, $h/L=0.02$ conditions and fixed propeller RPM of 538
The time histories of non-dimensional sinkage and trim in the tow condition and self-propulsion conditions with a fixed RPM of 538 are shown in Figure 9 for the three wave conditions. As noted earlier, the towing speeds used in the computations were taken from the self-propulsion speeds obtained for three wave-lengths conditions. The results indicate that the effects of the propeller action on the ship motion are marginal. It is seen that the oscillation amplitudes of the non-dimensional sinkage and trim increase with the wave length and height. Not surprisingly, the ship motion is greatly affected by the wave length and height.

![Figure 9: Time histories of the sinkage and trim in tow and self-propulsion conditions in head waves at λ/L=0.5, 1.0, 2.0, h/L=0.02 conditions and fixed propeller RPM of 538](image)

The time-averages of the speed loss, the propeller advance ratio, thrust and torque coefficients are presented in Table 6. As expected, the speed loss is the greatest at the resonance condition (λ/L=1.0). Due to the speed loss (lower ship speed), the advance ratios of the propeller in all wave conditions (computed using ship’s speed) are lower than that of the calm water condition with the lowest value occurring at the resonance condition. The propeller thrust and torque are higher in the presence of the waves with the highest value at λ/L=1.0.

![Table 6: Time averages of propeller advance ratio, thrust and torque coefficients, and percent change in time-averaged ship speed relative to the calm water condition](table)

| | $f$ | $k_T$ | $10k_Q$ | $\Delta\bar{u}\%$
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Calm water</td>
<td>1.151</td>
<td>0.194</td>
<td>0.06</td>
<td>0</td>
</tr>
<tr>
<td>$\lambda/L=0.5$</td>
<td>1.105</td>
<td>0.214</td>
<td>0.064</td>
<td>-4.43</td>
</tr>
<tr>
<td>$\lambda/L=1.0$</td>
<td>0.931</td>
<td>0.306</td>
<td>0.081</td>
<td>-19.46</td>
</tr>
<tr>
<td>$\lambda/L=2.0$</td>
<td>1.019</td>
<td>0.260</td>
<td>0.073</td>
<td>-12.22</td>
</tr>
</tbody>
</table>

The availability of the hull resistance, propeller thrust, and nominal wake from the computational results allow us to determine the propulsive parameters such as the thrust deduction and hull efficiency, whose variation with the wave length and height is plotted in Figure 10: $\lambda/L=0.0$ in this plot represents the calm water condition. The hull resistance and nominal wake-fraction were obtained from separate computations for the towed conditions.
4.2.3 Added Power

In the computations carried out to determine the added power, a PID controller was used to change the propeller RPM in order to maintain the desired ship speed of 1.11 m/s (equal to the ship speed for the calm water condition) and therefrom to see how much additional propulsive power will be needed.

Figure 11 shows the comparison of the time histories of the hull axial force ($C_{\text{Hull}}$) and the propeller thrust coefficients ($C_T$) for three wave conditions. Note that the propeller thrust coefficient here is non-dimensionalized using ship’s speed and wetted area in order to make the comparisons with the hull force coefficients meaningful. It is seen that the oscillation amplitudes of both coefficients increase with the wave length and height. While the time-averages of the propeller thrust and hull force should remain in equilibrium in a time-averaged sense, the oscillation amplitude of the hull axial force coefficient, at all wave conditions, is significantly larger than that of the propeller thrust. The oscillation frequencies of both forces are seen to coincide with the wave encounter frequency ($T_e$). However, there is clearly a phase difference between the two time histories at all wave conditions, which is presumably due to the complex nature of the forces involved (e.g., added-mass and damping forces on the hull, propeller force). It is worthwhile to remark in passing that the ship’s behavior depicted here would also be affected by the parameters selected for the PID controller.

The time histories of the pressure and viscous components of the hull axial force are shown in Figure 12. The oscillation amplitude of the pressure force coefficient is much larger than that of the viscous force, rapidly increasing with the wave length and height. The
oscillation amplitude of the viscous force coefficient peaks at $\lambda/L=1.0$ and falls back down at $\lambda/L=2.0$. Table 7 summarizes the time-averages of the pressure, viscous and total hull axial force coefficients for the calm water and the wave conditions. The time-averaged pressure force exhibits much larger variation than that of the viscous force coefficient.

![Figure 12](image1)

**Figure 12**: Time histories of non-dimensional pressure and viscous components of axial hull force in regular waves at $\lambda/L=0.5, 1.0, 2.0$, $h/L=0.02$ conditions and variable propeller RPM

**Table 7**: Time averages of non-dimensional pressure and viscous components and total of hull axial force coefficients ($\times 10^3$) at variable propeller RPM

<table>
<thead>
<tr>
<th>Wave condition</th>
<th>$C_{p,hull}$</th>
<th>$C_{vis,hull}$</th>
<th>$C_{hull}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calm water</td>
<td>0.836</td>
<td>3.548</td>
<td>4.384</td>
</tr>
<tr>
<td>$\lambda/L=0.5$</td>
<td>1.300</td>
<td>3.828</td>
<td>5.128</td>
</tr>
<tr>
<td>$\lambda/L=1.0$</td>
<td>5.497</td>
<td>4.603</td>
<td>10.100</td>
</tr>
<tr>
<td>$\lambda/L=2.0$</td>
<td>2.991</td>
<td>3.744</td>
<td>6.735</td>
</tr>
</tbody>
</table>

![Figure 13](image2)

**Figure 13**: Time histories of sinkage and trim in tow and self-propulsion conditions in regular waves at $\lambda/L=0.5, 1.0, 2.0$, $h/L=0.02$ conditions and variable propeller RPM

Figure 13 compares the time history of the non-dimensional sinkage and trim in waves in both the towed and the self-propulsion conditions with variable RPM. It is observed that the propeller action have a marginal effect on the ship motion. The similar conclusion was also drawn for the ship propelled at a constant RPM (see Figure 9). It is also seen that the oscillation amplitudes of both quantities increase with the wave length and height.

Table 8 shows the time-averages of the propeller advance ratio, thrust and torque coefficients, and the percentage change in the propeller RPM and propulsive power required to sustain the ship speed relative to the calm water condition. The propeller operates in a higher loading in waves (lower advance ratio) than in the calm water condition, generating higher thrust and torque. As expected, the highest propeller RPM and propulsive power are
required to maintain the ship speed in the resonance condition ($\lambda/L=1.0$). A similar trend was also observed for the propeller running at a constant RPM of 538 (see Table 6).

**Table 8:** Time averages of propeller advance ratio, thrust and torque coefficients, percent change in propeller RPM and power relative to calm water condition

<table>
<thead>
<tr>
<th>Wave condition</th>
<th>$\bar{\beta}$</th>
<th>$\bar{K}_T$</th>
<th>$10\bar{K}_Q$</th>
<th>$\Delta\bar{\beta}$ %</th>
<th>$\Delta\bar{P}$ %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calm water</td>
<td>1.151</td>
<td>0.194</td>
<td>0.060</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>$\lambda/L=0.5$</td>
<td>1.107</td>
<td>0.214</td>
<td>0.064</td>
<td>3.88</td>
<td>20.7</td>
</tr>
<tr>
<td>$\lambda/L=1.0$</td>
<td>0.940</td>
<td>0.296</td>
<td>0.080</td>
<td>22.18</td>
<td>145.7</td>
</tr>
<tr>
<td>$\lambda/L=2.0$</td>
<td>1.059</td>
<td>0.24</td>
<td>0.069</td>
<td>7.2</td>
<td>42.6</td>
</tr>
</tbody>
</table>

Figure 14 shows the time-histories of the nominal and the “total” wake fractions in the calm water and the wave conditions. Table 9 summarizes the time averages of the two quantities. Note that the total wake fraction was computed by monitoring a volume-averaged axial flow velocity immediately upstream of the propeller location. As such, it differs from the traditionally defined effective wake fraction. While the time-averaged nominal wake fraction does not noticeably change in presence of waves, its oscillation amplitude seems to increase with the wave length and height most likely due to the effects of the time-varying orbital velocity associated with the regular head waves. The orbital velocity changes its sign (direction) and magnitude in time depending on the axial location of the propeller relative to the wave crests and troughs. It is noteworthy that the total wake-fraction seems to widely vary with the wave length and height, significantly departing from the nominal wake. For the two longer (and larger) waves ($\lambda/L=1.0$ and $\lambda/L=2.0$), the time-averaged total wake fraction becomes negative, which implies the flow is accelerated enough – by the high propeller loading – to offset the velocity deficit due to the viscous wake.

**Figure 14:** Time histories of nominal (left) and total (right) wake fraction in calm water and regular waves conditions and variable propeller RPM

**Table 9:** Time average of nominal and total wake fraction at various flow conditions

<table>
<thead>
<tr>
<th>Wave condition</th>
<th>$W_n$</th>
<th>$W_{tot}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calm water</td>
<td>0.0650</td>
<td>0.064</td>
</tr>
<tr>
<td>$\lambda/L=0.5$</td>
<td>0.0657</td>
<td>0.021</td>
</tr>
<tr>
<td>$\lambda/L=1.0$</td>
<td>0.068</td>
<td>-0.084</td>
</tr>
<tr>
<td>$\lambda/L=2.0$</td>
<td>0.070</td>
<td>-0.066</td>
</tr>
</tbody>
</table>

The variation of the thrust deduction factor and the hull efficiency with the wave length is depicted in Figure 15. Interestingly enough, the hull efficiency has a distinctive minimum at
the resonance condition ($\lambda/L=1.0$). This finding is in good contrast to that for the hull efficiency obtained with a constant propeller RPM (Figure 10).

![Figure 15: Effects of wave length on thrust deduction factor and hull efficiency at variable propeller RPM condition](image)

**5 CONCLUSIONS**

- Coupled solutions of two-phase Unsteady Reynolds-Averaged Navier-Stokes equations and six degrees-of-freedom (6-DOF) rigid-body motion (RBM) equations are feasible for predicting the speed loss and added power for a self-propelling ship in regular head sea. We were able to robustly obtain the coupled URANS and 6-DOF RBM solutions for a 1/49 scaled model of the ONR Tumblehome (ONRT) (Model 5613) using unstructured grids.

- The speed loss in regular head seas can be predicted with a commendable accuracy. For the case of $\lambda/L = 1.0$ (resonance condition), the coupled URANS/RBM approach can predict the speed loss within 3.0%.

- Coupled solutions of the RANS and 6-DOF rigid-body motion equations allow us to predict not only the added resistance but also the speed loss and added power for a ship cruising in a regular head sea.

**AKNOWLEDGMENTS**

The authors would like to acknowledge support from the Department of Defense (DoD) High Performance Computing Modernization Program (HPCMP) under the Computational Research and Engineering Acquisition Tools and Environments (CREATE) Ship's Hydrodynamics Project, under the direction of Dr. Richard Vogelsong, current HPCMP CREATETM-SH Program Manager.

**REFERENCES**


DEVELOPMENT OF NUMERICAL METHOD TO SIMULATE FLOWS AROUND A SHIP IN REGULAR WAVES INCLUDING THE EFFECT OF SHIP PROPULSION PLANT MODEL

KUNIHIDE OHASHI

*National Maritime Research Institute
6-38-1 Shinkawa, Mitaka, Tokyo, Japan
e-mail: k-ohashi@nmri.go.jp

Key words: Ship Propulsion Plant Model, RANS, Overset-Grid Method

Abstract. A numerical method to simulate flows with propeller effects including the response of a ship propulsion plant has been developed. The dynamics of a ship propulsion plant is modeled by the function of the diesel engine control system. Propeller torque which is computed by the propeller model with the interaction of flow fields is put to the ship propulsion plant model, then the propeller rotational speed is obtained by solving the equation of the rotational motion of the propeller shaft line. Present method can reproduce the time history of propeller rotational speed and torque in the condition with the regular waves. The amplitude of fluctuations shows agreement with the measured data. The detail analysis of flow fields and ship motions which is difficult to be obtained at the experiment is also carried out.

1 INTRODUCTION

Recently, Reynolds Averaged Navier-Stokes(RANS) simulations are utilized at the design stage of ship performance, and the numerical simulations are gradually applied to the more complex problems. URANS solver which can cope with the overset-grid method is coupled with a ship propulsion plant model through a propeller model which compute the body forces which express the propulsion effect in the present research. The present method can treat the response of the ship propulsion plant and the fluctuation of the propeller torque with ship motions in waves. The ship propulsion plant model[1] based on the mathematical equation of diesel engine components is employed. Computational results are validated with the experimental data[2]. Additionally, the detail analysis for the flows around the hull with motions are carried out. The relation between the response of the ship propulsion plant and flow fields are examined.
2 COMPUTATIONAL METHOD

2.1 Base solver

An in-house structured CFD solver[3] is employed. The governing equation is 3D RANS equation for incompressible flows. Artificial compressibility approach is used for the velocity-pressure coupling. Spatial discretization is based on a finite-volume method. A cell centered layout is adopted in which flow variables are defined at the centroid of each cell and a control volume is a cell itself. Inviscid fluxes are evaluated by the third-order upwind scheme based on the flux-difference splitting of Roe. The evaluation of viscous fluxes is second-order accurate. For unsteady flow simulations, a dual time stepping approach is used in order to recover incompressibility at each time step. It is consisted from the second order two-step backward scheme for the physical time stepping and the first order Euler implicit scheme for the pseudo time. The linear equation system is solved by the symmetric Gauss-Seidel (SGS) method.

For free surface treatment, an interface capturing method with a single phase level set approach is employed.

Incoming Regular headsea waves are generated at the region inside of the computational domain[4]. Body motions are obtained by solving the equations of motion, and motions are taken into account by a moving grid technique with the grid deforming methodology. Grid velocities are contained in the inviscid terms to satisfy the geometrical conservation law. The grid velocities are derived from the volume where an each cell face sweeps. The boundary condition on a body is given as the velocities of the body motion.

The regions where the overset relations are composed deform with the body motions to maintain the overset information, and the amount of deformations gradually decreases with the distance from the body surfaces. Such the way is adapted to be able to avoid the computational load with using the dynamic overset-grid method.

2.2 Overset-grid method

The weight values for the overset-grid interpolation are determined by an in-house system[5]. The detail of the system can be found on [5], the summary is described.

1. The priority of the computational grid is set.

2. The cells of a lower priority grid and inside a body is identified (called as in-wall cell in here).

3. Receptors cells which the flow variables have to be interpolated from donor cells are defined. Two cells on a higher priority grid and facing to the outer boundary are set as receptor cells to satisfy the third order discretization of NS solver. Additionally, two cells neighborhood of in-wall cells, the cells of a lower priority grid and inside the domain of a higher priority grid are also set as the receptor cell.

4. The weight values for the overset interpolation are determined by solving the inverse problem based on Ferguson spline interpolation.

Flow variables of the receptor cell are updated when the boundary condition is set. The forces and moments are integrated on the higher priority grid to eliminate the lapped region on body
surfaces. At first, the cell face of the lower priority grid is divided into small pieces. Secondly, the small piece is projected to the cell face of the higher priority grid by using the normal vector of the higher priority face. Then the 2D solid angle is computed and the small piece is decided in or out of the higher priority face. Once the small piece is in the higher priority face, the area ratio of the piece is set to zero. Finally, the area ratio is integrated on the lower priority face, then we have the ratio to integrate the forces and moments on lower priority face.

2.3 Propeller model

The propeller model based on the potential theory\cite{6} is applied to archive the propulsion condition. The propeller effect is taken account by the body forces which are computed by the following equations.

\[
f_x(r, \theta) = \frac{\Gamma(r, \theta)V_\theta}{r} - \frac{1}{2}C_D N c(r) \sqrt{1 + \left(\frac{h(r)}{r}\right)^2 V_{ox} V_{o\theta}} \frac{V_{ox}}{2\pi r}
\]

\[
f_t(r, \theta) = \frac{\Gamma(r, \theta)V_x}{r} + \frac{1}{2}C_D N c(r) \sqrt{1 + \left(\frac{h(r)}{r}\right)^2 V_{ox}^2} \frac{V_{o\theta}}{2\pi r}
\]

\[f_y = f_t \cos \theta, \quad f_z = f_t \sin \theta\]

where \(r\) and \(\theta\) are the cylindrical coordinate at the propeller plane. The propeller circle is divided into the fan-shaped segments at \((r, \theta)\). \(\Gamma(r, \theta)\) is the vortex strength, and \(\Gamma(r, \theta)\) is determined by the equation which is based on the boundary condition at the segment \((r, \theta)\). \(V_x, V_\theta\) are the inflow velocities to the segment, \(V_{ox}, V_{o\theta}\) are the circumferential averaged velocities. \(C_D\) is a drag coefficient which is given by the empirical formula, \(N\) is the blade number of a propeller, \(c(r)\) is a chord length at each radial direction, \(h(r)\) is the pitch of a free stream vortex.

The coupling between the flow field and the propeller model is made by the velocities \(V_x, V_\theta\). At first, the velocity at the segment center is interpolated from the computational grid. Then, the \(\Gamma(r, \theta)\) is determined, and body forces are computed. Finally, the cross section between the computational grid and the propeller segment is searched, and the forces at the cell is derived by the multiplication of body forces and the cross section area. The coordinate transfer which the body fixed coordinate to the earth fixed coordinate or its inverse are made by using Euler angles.

2.4 Ship Propulsion Plant Model

The ship propulsion plant model\cite{1} based on the mathematical equation of diesel engine components is employed. The model can simulate the behavior of the diesel engine in the transient conditions interacting with the major components of the ship propulsion plant.

At first, the speed sensor gain is determined by the sensor model.

\[\bar{Z}_s = K_s (\bar{n}_{sp} - \bar{n}_e)\]

where \(K_s\) is speed sensor gain coefficient, \(\bar{n}_{sp}\) is target engine rotational speed, \(\bar{n}_e\) is actual speed of engine.

The next equation is the summation unit, and \(\bar{Z}_{ip}\) is isodromic feedback amount, \(\bar{Z}_{fb}\) is proportional feed back amount.
\[ \ddot{\psi} = \ddot{Z}_s - \ddot{Z}_{ip} - \ddot{Z}_{fb} \]  
\hspace{1cm} (5)

Dead band is applied by following equation.

\[ \ddot{Z}_0 = \begin{cases} 
\dot{\psi} - \epsilon/2 & \text{if } \dot{\psi} > \epsilon/2 \\
0 & \text{if } |\dot{\psi}| \leq \epsilon/2 \\
\dot{\psi} + \epsilon/2 & \text{if } \dot{\psi} < -\epsilon/2 
\end{cases} \]  
\hspace{1cm} (6)

where \( \epsilon \) is dead band.

The fuel flow rate is determined by the power piston model.

\[ T_{pp} \frac{d\bar{h}_p}{dt} = \bar{Z}_0 \]  
\hspace{1cm} (7)

where \( T_{pp} \) is the time constant of the power piston.

Eq.(8) and eq.(9) are the feedback mechanism model to determine the isodromic feedback gain \( \bar{Z}_{ip} \) and the proportional feedback gain \( \bar{Z}_{fb} \).

\[ T_i \frac{d\bar{Z}_{ip}}{dt} + \bar{Z}_{ip} = K_i T_i \frac{d\bar{h}_p}{dt} \]  
\hspace{1cm} (8)

\[ \bar{Z}_{fb} = K_{fb} \bar{h}_p \]  
\hspace{1cm} (9)

where \( T_i \) is the time constant of the isodromic feedback, \( K_i \) is the gain coefficient of isodromic feedback, \( K_{fb} \) is gain coefficient of the proportional feedback.

The engine torque \( \bar{Q}_e \) can be computed by the equation based on the relationship between the fuel flow rate \( \bar{h}_p \) and engine revolution number \( \bar{n}_e \).

\[ \bar{Q}_e = 0.5\bar{h}_p^{2/3} + 1.5\bar{h}_p^{1/3}\bar{n}_e - \bar{n}_e^2 \]  
\hspace{1cm} (10)

The variables \( \bar{n}_e, \bar{h}_p, \bar{Q}_e \) are non-dimensionalized by the engine revolution number \( n_{MCR} \), the fuel flow \( h_{MCR} \) and the torque \( Q_{eMCR} \) at the Maximum Continuous Rating(MCR)

\[ \bar{n}_e = n_e/n_{MCR}, \hspace{0.2cm} \bar{h}_p = h_p/h_{MCR}, \hspace{0.2cm} \bar{Q}_e = Q_e/Q_{eMCR} \]  
\hspace{1cm} (11)

Finally, the shaft rotational speed is obtained by solving the equation of the rotational motion of a shaft line.

\[ 2\pi I_p \frac{dn}{dt} = Q_e - Q_p \]  
\hspace{1cm} (12)

where \( I_p \) is moment of inertia of whole shaftline, \( Q_p \) is propeller torque.

The equations of the ship propulsion plant model are discretized by the first order. The propeller torque \( Q_p \) is obtained by the simplified propeller model. The actual speed of engine \( n_e \) equals to the propeller rotational speed \( n \) in the similar way of the measurement condition.
3 COMPUTED RESULTS

The container hull form which is utilized on the experiment[2] is selected. The model ship length is about 4.0m and Reynolds number is \( R = 2.8 \times 10^6 \). Froude number is \( Fn = 0.128 \). The particular of the propeller is same with the experiment. The wave length ratio of incoming regular headsea waves is \( \lambda/L = 0.9, 1.0, 1.1 \), and the wave height ratio \( h_w/L \) is 0.025. The motions are free to pitch and heave. The non-dimensional physical time step size is set to \( \Delta t = 0.002 \).

Table 1 shows the division number of computational grids arranged with the priority of the overset-grid method. Figure 1 shows the grids near the hull body. The minimum spacing on the wall is set as \( y^+ = 1 \) to apply \( k-\omega \) SST turbulence model. The hull and refined rectangular grid are overlapped to the rectangular grid to generate the incoming regular headsea waves.

<table>
<thead>
<tr>
<th>Grid</th>
<th>IM×JM×KM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Refined Rect.</td>
<td>65×65×65</td>
</tr>
<tr>
<td>Hull</td>
<td>193×217×65</td>
</tr>
<tr>
<td>Rect.</td>
<td>217×105×57</td>
</tr>
</tbody>
</table>

Table 1: Division number of computational grid

Figure 1: Computational grids

Table 2 shows coefficients and time constants of ship propulsion plant model, target engine rotational speed. All the coefficients and time constants are used at the experiment and non-dimensionalized.

Figure 2 shows the comparison of time history of propeller rotational speed, Figure 3 shows the comparison of time history of propeller torque, and Figure 4 depicts the comparison of the non-dimensionalized fuel consumption. All the figures are at the condition \( \lambda/L = 1.1 \), the propeller rotational speed and propeller torque are dimensionalized for the direct comparison with the experimental data. As similar with the experiment data, the time phase is corrected.
Table 2: Coefficients and time constants of ship propulsion plant model

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$K_s$</td>
<td>12.5</td>
</tr>
<tr>
<td>$T_{pp}$</td>
<td>$2.771 \times 10^{-3}$</td>
</tr>
<tr>
<td>$K_i$</td>
<td>1.4</td>
</tr>
<tr>
<td>$T_i$</td>
<td>$5.773 \times 10^{-2}$</td>
</tr>
<tr>
<td>$K_{fb}$</td>
<td>0.75</td>
</tr>
<tr>
<td>$n_{MCR}$</td>
<td>75.05</td>
</tr>
<tr>
<td>$n_{sp}$</td>
<td>50.0</td>
</tr>
<tr>
<td>$Q_{MCR}$</td>
<td>$4.573 \times 10^{-6}$</td>
</tr>
<tr>
<td>$I_p$</td>
<td>$2.655 \times 10^{-10}$</td>
</tr>
</tbody>
</table>

by the propeller rotational speed to start from the trough of fluctuations. The computational results can reproduce the fluctuations of the variables. The differences of the time averaged value between the measurement and computation are mainly caused by the difference of the time averaged value of the inflow to the propeller. The computational results are able to simulate the characteristics which the propeller rotational speed is decreasing to the contrast of the propeller torque, additionally the fuel consumption is increasing to maintain the target engine rotational speed.

![Figure 2: Comparison of time history of propeller rotational speed](image1)

![Figure 3: Comparison of time history of propeller torque](image2)

From Figure 5 to Figure 7 show the comparisons of the double amplitude with changing $\lambda/L$. The computational results show agreement with the measured results. The double amplitude when the propeller rotational speed is fixed at the time averaged value with the ship propulsion...
The present computational result can simulate the decreasing of the amplitude with the ship propulsion plant model.

Figure 4: Comparison of time history of fuel consumption

Figure 5: Comparison of double amplitude of propeller rotational speed

Figure 6: Comparison of double amplitude of propeller torque

Figure 7: Comparison of double amplitude of fuel consumption

Figure 8 shows the time histories of the propeller rotational speed, ship motions and wave
height within the one encounter period at \( \lambda/L = 1.1 \) condition. The propeller rotational speed closely fluctuates with the heave motion of the ship. Figure 9 shows the free surface near the hull from \( T_1 \) to \( T_4 \) in Figure 8, and Figure 10 shows the axial velocity contour and cross flow vectors at the propeller plane in ship fixed coordinate.

T1 The heave motion reaches maximum value, and the pitch motion takes bow up position. The ship nose appears from the free surface, and inflow velocity \( w \) to the propeller takes negative value. The propeller rotates in clockwise direction seeing from the stern, thus the axial velocity \( u \) of the starboard side is faster than the velocity of the port side at a calm water condition. On the contrary, the axial velocity of the port side is faster than the velocity \( u \) of the starboard side at the time \( T_1 \).

T2 The heave motion becomes negative value, and pitch motion takes bow down position. The inflow velocity \( w \) to the propeller still takes negative value. The magnitude of the velocity \( w \) is smaller than the value of the time \( T_1 \). The axial velocity \( u \) is accelerated in both sides.

T3 The heave motion reaches minimum value, and the pitch motion takes from bow down position to the bow up position. The inflow velocity \( w \) to the propeller takes positive value. The axial velocity \( u \) is accelerated in the starboard side.

T4 The heave motion takes almost zero position, and the pitch motion takes bow up position again. The ship nose also appears from the free surface. The inflow velocity \( w \) to the propeller still takes positive value. The magnitude of the velocity \( w \) is smaller than the value of the time \( T_3 \). The axial velocity \( u \) is accelerated as similar with the calm water condition.

The inflow velocities are changed by the incoming waves and ship motions, then the propeller torque which is the major input value to the ship propulsion plant model varies. The propeller revolution number is changed with the behavior of the ship propulsion plant model. The interactions between the change of the propeller characteristics in waves and the vary of the propeller revolution number which is caused by the behavior of the ship propulsion plant model is major reason for the time fluctuation of the variables.

4 CONCLUSIONS

- The numerical method including the behavior of ship propulsion plant model by using the URANS solver and the propeller model has developed.

- From the comparisons with the experimental data, the present method can reproduce the time fluctuations of the fuel flow rate, the propeller torque and rotational speed.

- The detail flow analysis including the effect of the ship propulsion plant model is become possible by the present method.

5 ACKNOWLEDGEMENT

This work has been supported by JSPS KAKENHI Grant Number JP16K06919.
Figure 8: Time history of propeller rotational speed and motions

Figure 9: Instantaneous view of free surface and ship with ship propulsion plant model

REFERENCES


Figure 10: Axial velocity contour and cross flow vectors with ship propulsion plant model


Estimation of Added Wave Resistance with CFD Code

Romain Huret*, Doriane Causeur†, Anne-Sophie Dubois‡, Aurélien Drouet†, Olivia Thilleul°

* DCNS Research/SIREHNA
Technocampus OCEAN
5 rue de l’Halbrane, 44340 Bouguenais, France
Email: romain.huret@sirehna.com - Web page: http://www.sirehna.com

† HYDROCEAN
8 Boulevard Albert Einstein, 44000 Nantes, France - Web page : http://www.hydrocean.fr
Email: doriane.causeur@hydrocean.fr, aurelien.drouet@hydrocean.fr

‡ STX France SA
Avenue Bourdelle CS 90180, 44613 Saint-Nazaire, France
Email: anne-sophie.dubois @stxeurope.com - Web page: http://www. stxeurope.com

° IRT Jules Verne
Chemin du Chaffault, 44340 Bouguenais, France
Email: olivia.thilleul@irt-jules-verne.fr – Web page: http://www.irt-jules-verne.fr/

Key words: Computational Methods, Wave, Sea Keeping

This paper is part of the Bassin Numérique project managed by IRT Jules Verne (French Institute in Research and Technology in Advanced Manufacturing Technologies for Composite, Metallic and Hybrid Structures). The authors wish to associate the industrial and academic partners of this project; respectively DCNS Research/SIREHNA, HYDROCEAN, STX France, Bureau Veritas.

Abstract. A numerical towing tank needs to efficiently estimate the ship performances in both calm water and in regular waves. The knowledge of ship performances are mandatory during the design phases in order to provide architects with values helping in technological choices. In this context, numerical towing tanks appear as a more versatile solution than time consuming and costly model tests. The French Technical Research Institute IRT Jules Verne conducts studies to assess and validate methodologies based on CFD simulations to evaluate added resistance in regular waves.

The present work conducted in the "Bassin Numérique" project provides a preliminary sensitivity analysis which aims to validate the numerical settings necessary to model the wave propagation. The main result of this preliminary study enables to specify accurate meshes for wave propagation.

The present paper focuses on the validation study done by three different members of the IRT Jules Verne using three CFD solvers on four test cases: one static vertical cylinder [1] and three ships in head wave condition [2], [3] and [4]. For each case, numerical results are compared with towing tank experiments in terms of added resistance and motions.

The different wave conditions and test cases allow covering the wide range of encountered...
wave frequencies and dealing separately with the cases of diffraction at zero speed, diffraction with forward speed and finally including radiation. Most of the results correctly fit the experimental data, especially in terms of heave and pitch. The added resistance is also accurately simulated for sufficiently high wave lengths.

1 FLOW SOLVERS AND MESHES

For the purpose of the study, three flow solvers are used to investigate different tests cases. These three solvers are STAR-CCM+, ISIS-CFD, NavalFOAM. They solve the RANSE (Reynolds Average Navier-Stokes Equations) by mean of Finite Volume methods. These solvers allow setting simulations in various ways for examples by choosing the linear solver algorithms the discretization schemes… The main settings used are briefly presented below.

The boundary conditions used are:
- Inlet velocity, or wave generation condition upstream,
- Zero pressure gradients downstream
- Slip wall on the bottom of the domain
- Hydrostatic pressure on the top of the domain
- Symmetry condition on the side of the domain
- No slip, wall function on the ship

Turbulence is modeled with the common two equations eddy-viscosity formulations k-ω SST. This closure model predicts the turbulence by means of the kinematic turbulent intensity (k) and the turbulent specific dissipation rate (ω). Wall functions are applied on the ship hull except deck which is almost always set as a slip wall (i.e. no friction).

Because of different implementations in the solvers, the wave is either a 2nd order Stokes wave (ISIS-CFD) or a 5th order Stokes wave (STAR-CCM+ and NavalFOAM). The free-surface is modeled by the tracking method VoF (Volume of Fluid). With this method waves can be reflected by the outlet back in the domain. This automatically leads to wrong and non-converged solution. To overcome this problem the same kind solution is used with all solvers. It consists in adding a damping source term in the momentum equation to reduce to zero the vertical component of the velocity due to the wave at the outlet. The zero gradient boundary condition can therefore be respected at the outlet. In case of free trim and sinkage simulation, the ship motion is taken into account by deforming the mesh using the morpher included in the solvers.

The meshes are generated by the recommended tools of each solver: the integrated mesh generator is used for STAR-CCM+, Hexpress provided by Numeca is used for ISIS-CFD; snappyHexMesh is used for NavalFOAM.

2 TEST CASES

The four test cases used in this study have been chosen for their wide range of sea states (Table 1) and because of the different experimental data available.
The first test case is a fixed cylinder in regular wave [1]. This is an academic case with experimental data from the wave basin of ECN (Ecole Centrale de Nantes). The waves used for this case have the greatest camber among the entire waves used in this study. A mesh and time step sensitivity analysis is realized for this set up, the final results are presented in next section. For this diffraction case the water elevation is measured experimentally at three positions around the cylinder, and the first three harmonics of drag force are also available.

The second test case is the frigate DTMB in regular head wave either fixed or free to heave and pitch (experimental data came from IIHR). This case is of main interest because there are various experimental data available, including flow measurements. Also measurements at different speeds and wave elevations for similar wave periods are available which allow validating the mesh and solvers for linear extrapolation. Unfortunately this case does not contain a lot of data concerning resistance; to overcome this issue the next two cases have been treated.

The third and fourth test cases are respectively KCS for which experimental study has been conducted by FORCE Technology, and KVLCC2 in regular head waves, both free to heave and pitch. These cases are amongst the most recent test cases and are accurate. For these two cases only, motions and forces have been post-processed.

<table>
<thead>
<tr>
<th></th>
<th>Cylinder</th>
<th>DTMB</th>
<th>KCS</th>
<th>KVLCC2</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\lambda/L_{pp}$ [%]</td>
<td>8.09</td>
<td>0.50 to 1.50</td>
<td>0.65 to 2.75</td>
<td>0.50 to 2.00</td>
</tr>
<tr>
<td>$A_k = \pi H/\lambda$ [%]</td>
<td>14.7 and 21.5</td>
<td>2.5 to 7.5</td>
<td>4.7 to 5.2</td>
<td>1.6 to 6.3</td>
</tr>
</tbody>
</table>

*Figure 1: Non dimensional sea states distribution*
Table 2: Body mains characteristics used in simulations

<table>
<thead>
<tr>
<th></th>
<th>Cylinder</th>
<th>DTMB</th>
<th>KCS</th>
<th>KVLCC2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scale</td>
<td>1:1</td>
<td>1:46.588</td>
<td>1:37.890</td>
<td>1:100</td>
</tr>
<tr>
<td>Froude number</td>
<td>Fr [-]</td>
<td>0</td>
<td>0.28</td>
<td>0.41</td>
</tr>
<tr>
<td>Length between perpendicular</td>
<td>Lpp [m]</td>
<td>0.625</td>
<td>3.048</td>
<td>6.070</td>
</tr>
<tr>
<td>Length at water line</td>
<td>Lwl [m]</td>
<td>-</td>
<td>3.052</td>
<td>6.136</td>
</tr>
<tr>
<td>Beam</td>
<td>Bwl [m]</td>
<td>-</td>
<td>0.409</td>
<td>0.850</td>
</tr>
<tr>
<td>Draft</td>
<td>T [m]</td>
<td>0.938</td>
<td>0.132</td>
<td>0.285</td>
</tr>
<tr>
<td>Volume</td>
<td>V [m³]</td>
<td>0.288</td>
<td>0.083</td>
<td>0.957</td>
</tr>
<tr>
<td>Wetted Surface</td>
<td>Sw [m²]</td>
<td>2.148</td>
<td>1.371</td>
<td>6.610</td>
</tr>
<tr>
<td>Longitudinal position of CoG From aft PP</td>
<td>LCG [m]</td>
<td>1.539</td>
<td>1.539</td>
<td>2.855</td>
</tr>
<tr>
<td>Vertical position of CoG From keel</td>
<td>VCG [m]</td>
<td>0.132</td>
<td>0.378</td>
<td>0.109</td>
</tr>
</tbody>
</table>

3 RESULTS

The results presented in this section are the final submitted results of the three members of the project. The two first cases have been treated by each member, while the two last are treated by only one different participant for each case.

For each test case, the post processing of the different signal (either force or motion) is done by fitting, with a least square method, a sum of harmonically related sinusoids (1) over the three last periods of the signal. For the wave elevation only, the fitting is done against the analytical formulation of a 2nd order Stokes wave in infinite depth (2).

\[ h(t) = \sum_{n=0}^{3} a_n \cos(2\pi n t + \phi_n) \]  
\[ \eta(x,t) = \frac{H}{2} \left[ \cos\left(\frac{\pi}{\lambda} \left( x - \frac{t}{T} \right) \right) + \frac{1}{2} \frac{\pi H}{\lambda} \cos\left( 4\pi \left( \frac{x}{\lambda} - \frac{t}{T} \right) \right) \right] \]

During the realization of the cylinder test case the meshes, time step, and damping zone are investigated in order to obtain minimum error between the simulated wave elevation and the analytical formulation of a Stokes wave, either without the cylinder or with it by taking care of checking waves in an area without diffracted waves.

The cell sizes are varied from 30 to 150 cells per wave length in the direction of propagation and from 5 to 30 cells per wave height in vertical direction. For the lateral direction the cell size should not be more than twice the size in the propagation direction. At last the aspect ratio of the cells (dx/dz) appears to be more important than the actual discretization of the mesh. That is to say, if aspect ratio is too large then the error between the wave elevation and the analytical formulation will be too large even if the longitudinal and vertical discretizations are both correct. This limitation is not the same for each solver.
During this mesh sensitivity analysis, the time step is varied from 50 to 300 time steps per wave period. As expected it appears that the accuracy of the computation is directly linked to the Courant number in the vicinity of the free surface. As for the cell size the limit is not equal for each solver. In STAR-CCM+ the HRIC formulation is used for the convective part of the free surface transport equation. With this scheme the limit due to the blending formulation on the Courant number is set to 0.7. In ISIS-CFD, the BRICS scheme is used. For this one the limit is of 0.3. Both this limitation and a sufficiently large number of time steps per wave period have to be fulfilled in order to correctly propagate the wave.

At last, results for the cylinder are presented in (Table 3, Table 4 and Figure 2). For the two wave heights, the first harmonic of the force signal is similar between simulations and experiments the error is below 1% for 6 simulations out of 9. Besides, the results for the second and third harmonics seem not well predicted. This large error is mostly due to the low values which are compared.

Most of the settings used for the cylinder test case are used for the next three test cases. Time steps are reduced because of the ship motion: the time step discretization is based on the encountered frequency instead of using a discretization per wave period.

For the DTMB 5512 test cases in fixed position, the free-surface and the velocity field at the propeller location are exported each quarter of wave periods. Results are presented on figures 3 and 4. Both the free-surface elevation and the wave patterns are comparable between simulations and experiments. Also the solvers provide really similar results even with different meshes.

The motions simulated for the case DTMB 5512 free to heave and pitch are quite similar to the experimental results. For most of the wave lengths on Figure 5 and Figure 6 there are three simulations and experiments, one for each wave height. These graphs show similar results between CFD and experiments either for the amplitude of the motions and for the phase of the signal.

Finally, on Figure 7 and Figure 8, the transfer functions of motions and added resistance (3) are represented, for respectively the KCS and the KVLCC2 in head waves. In these two cases the three transfer functions fit well between experiments and simulations for the wave lengths close and above the wave length of the maximum added resistance. For smaller wave lengths the added resistance is less similar while motions are still well captured.

Most of these results show good agreement between any CFD solvers and experiments. This is especially satisfying as CFD and experiments match well for various data such as wave elevation, velocity, motions and forces. Also the three solvers provide quite similar data which increases confidence in these tools for solving problems related to waves.

\[ ARTF = \frac{F_{\text{wave}} - F_{\text{calm\_water}}}{\rho g (H / 2)^2 B^2 / Lpp} \]
4 CONCLUSIONS AND PERSPECTIVES

The study presented in this paper consists of comparisons between the predictions of regular wave interaction with ships, of three CFD codes (STAR-CCM+, ISIS-CFD, and NavalFOAM) against experimental data. The comparisons are done on four test cases: one fixed cylinder and three ships (DTMB, KCS, and KVLCC2) free to heave and pitch in head waves. The numerous experimental data allows setting and validating the methodologies on different aspects of sea keeping, such as wave elevation, motion of ship in waves, and added resistance.

The CFD results presented in this study are mostly in good agreement with experimental data. There are further works to do on small wave lengths modelling for both KCS and KVLCC2 test cases. For these configurations the mesh density has to be increased dramatically while time steps have to be reduced, which leads to an increase of computations cost, and therefore it has not been possible to correctly model these configurations yet.

This validation study was the first step toward a wider use of CFD solvers in the sea keeping field. This work and the following are intended to use CFD in order to answer several questions such as which is the added resistance of ship in a particular sea state, which hull is the most efficient, what are the loads on the hull and superstructures.

There are numerous possibilities to pursue this work. Amongst them will be the investigation of the effects of appendages, and the scale effects which are of great interest to increase our confidence in full scale computations. Moreover the response of ship in irregular sea states has to be investigated. For this purpose the use of SWENSE solvers seems to be a very promising solution. Another work will be developing CFD methodologies for extreme sea states configurations.

REFERENCES


Figure 2: Wave elevation around the cylinder – T=1.8s – H=0.237m

Table 3: Forces on the cylinder – T=1.8s – H=0.237m

<table>
<thead>
<tr>
<th></th>
<th>EFD</th>
<th>DCNS</th>
<th>HO STAR-CCM+</th>
<th>HO ISIS</th>
<th>STX STAR-CCM+</th>
<th>STX ISIS</th>
<th>STX NavalFOAM</th>
</tr>
</thead>
<tbody>
<tr>
<td>F1</td>
<td>487 N</td>
<td>-0.2%</td>
<td>-4.5%</td>
<td>0.9%</td>
<td>-0.3%</td>
<td>0.3%</td>
<td></td>
</tr>
<tr>
<td>F2</td>
<td>19.8 N</td>
<td>-12.5%</td>
<td>-19.1%</td>
<td>-59.7%</td>
<td>-27.1%</td>
<td>-29.4%</td>
<td></td>
</tr>
<tr>
<td>F3</td>
<td>8.2 N</td>
<td>30.4%</td>
<td>14.9%</td>
<td>47.9%</td>
<td>42.6%</td>
<td>36.8%</td>
<td></td>
</tr>
</tbody>
</table>

Table 4: Forces on the cylinder – T=1.8s – H=0.346m

<table>
<thead>
<tr>
<th></th>
<th>EFD</th>
<th>DCNS</th>
<th>HO STAR-CCM+</th>
<th>HO ISIS</th>
<th>STX STAR-CCM+</th>
<th>STX ISIS</th>
<th>STX NavalFOAM</th>
</tr>
</thead>
<tbody>
<tr>
<td>F1</td>
<td>709 N</td>
<td>-3.1%</td>
<td>-6.0%</td>
<td>-0.1%</td>
<td>0.1%</td>
<td></td>
<td></td>
</tr>
<tr>
<td>F2</td>
<td>24.3 N</td>
<td>-0.1%</td>
<td>1.8%</td>
<td>-13.4%</td>
<td>-89.6%</td>
<td></td>
<td></td>
</tr>
<tr>
<td>F3</td>
<td>24.2 N</td>
<td>14.2%</td>
<td>15.2%</td>
<td>23.2%</td>
<td>16.4%</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Figure 3: Comparison of wave elevation - DTMB
Figure 4: Comparison of velocity level – DTMB
Figure 5: Comparison of heave and pitch in wave – DTMB - Fr=0.28

Figure 6: Comparison of heave and pitch in wave – DTMB - Fr=0.41
Figure 7: Comparison of added resistance in wave – KCS

Figure 8: Comparison of added resistance in wave – KVLCC2
INFLUENCE OF THE DRAFT TO SHIP DYNAMICS IN THE VIRTUAL TANK BASED ON OPENFOAM

P. DU*, A. OUAHSINE*, P. SERGENT†

* Laboratoire Roberval, UMR-CNRS 7337
Sorbonne Universités, Université de Technology de Compiègne
Centre de Recherches Royallieu, CS 60319, 60203 Compiègne cedex, France
e-mail: pp1565156@126.com (P. Du); ouahsine@utc.fr (A. Ouahsine)

† CEREMA-134, rue de Beauvais, CS 60039, 60200 Compiègne, France

Key words: Draft, KCS, OpenFOAM, RANS

Abstract. A virtual tank is built based on OpenFOAM. The mesh is created using the ‘blockMesh’ and the ‘snappyHexMesh’ utilities successively. The ‘interDyMfoam’ solver is used to solve the flow fields. The results are compared with the experimental data of the Tokyo 2005 CFD workshop, which show good agreement. The cases with seven different drafts are further simulated. It is found that a small draft can make the ship unstable and weaken the effect of the bulbous bow.

1 INTRODUCTION

For a ship, draft is the vertical distance from the keel to the waterline. In the shipping industry, it is an important criteria for calculating the ship displacement and determining the water depth that a ship can safely navigate. Normally, the ship resistance declines with the decrease of the draft. To achieve a good hydrodynamic performance, a bulbous bow is always designed in front of the ship to modify the water flows around the ship, reduce drag and increase speed, fuel efficiency and stability. However, its effect is only optimized when it is submerged underwater. Besides, a small draft will make the propeller operate near the free surface or even in the air. Therefore the draft of a ship should be controlled properly in real maneuvering. In this paper, a virtual tank is designed based on OpenFOAM [1] and a KCS (KRISO Container Ship) is tested with different drafts. The results are validated using the experimental data of the Tokyo 2005 CFD workshop [2]. The influences of drafts to the ship are finally investigated and concluded.
2 NUMERICAL METHODS

2.1 Governing equations

The incompressible URANS (Unsteady Reynolds-Averaged Navier-Stokes) equations for two-phase flow can be written as [3, 4, 5]:

\[ \nabla \cdot \mathbf{U} = 0 \] \hspace{1cm} (1)

\[ \frac{\partial \alpha}{\partial t} + \nabla \cdot [\alpha (\mathbf{U} - \mathbf{U}_g)] + \nabla \cdot [\alpha (1 - \alpha) \mathbf{U}_r] = 0 \] \hspace{1cm} (2)

\[ \frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot [\rho (\mathbf{U} - \mathbf{U}_g) \mathbf{U}] = -\nabla p_{\text{rgh}} - \mathbf{g} \cdot \mathbf{x} \nabla \rho + \nabla \cdot (\mu_{\text{eff}} \nabla \mathbf{U}) + (\nabla \mathbf{U}) \cdot \nabla \mu_{\text{eff}} + \mathbf{f}_\sigma \] \hspace{1cm} (3)

where \( \mathbf{U} \) is the velocity field, \( \mathbf{U}_g \) is the grid velocity considering the mesh motion. \( p_{\text{rgh}} = p - \rho \mathbf{g} \cdot \mathbf{x} \) is a modified pressure defined in OpenFOAM. \( \mathbf{g} \) is the gravitational acceleration. \( \mu_{\text{eff}} = \rho (\nu + \nu_t) \) is the effective dynamic viscosity, where \( \nu \) and \( \nu_t \) are the kinematic and eddy viscosity respectively. \( \nu_t \) is obtained from a specific turbulence model. In this study, the SST \( k-\omega \) model is adopted. The pressure velocity coupling is realized using the PIMPLE algorithm.

Equation 2 is the VoF (Volume of Fluid) equation with the artificial compression. \( \mathbf{U}_r = \mathbf{U}_l - \mathbf{U}_g \) is the relative velocity between two phases. The subscripts 'l' and 'g' designate 'liquid' and 'gas' phases. \( \alpha \) is the volume fraction defined as [6, 7, 8]:

\[
\begin{align*}
\alpha &= 0 \text{ gas} \\
0 < \alpha < 1 \text{ interface} \\
\alpha &= 1 \text{ liquid}
\end{align*}
\] \hspace{1cm} (4)

\( \mathbf{f}_\sigma = \sigma \kappa \nabla \alpha \) is the surface tension term, where \( \kappa \) is the mean curvature of the free surface, determined using the equation:

\[ \kappa = -\nabla \cdot \left( \frac{\nabla \alpha}{|\nabla \alpha|} \right) \] \hspace{1cm} (5)

The physical properties (density, dynamic viscosity) are calculated as weighted averages based on the phase fraction:

\[
\begin{align*}
\rho &= \alpha \rho_l + (1 - \alpha) \rho_g \\
\mu &= \alpha \mu_l + (1 - \alpha) \mu_g
\end{align*}
\] \hspace{1cm} (6)

2.2 Mesh generation

The mesh is generated using the 'blockMesh' and 'snappyHexMesh' utilities in OpenFOAM. 'blockMesh' is used to generate the background mesh and 'snappyHexMesh' is used to generate the mesh of the ship. The non-dimensional wall distance \( y^+ \) of the boundary layer mesh in this study is in the range of 30 < \( y^+ \) < 60.

As shown in Fig.1, a half domain is created using the 'symmetry' boundary condition to decrease the mesh number. The region of the ship, especially the bow and the aft is refined to better capture the flow fields. The final mesh has about 1,000,000 cells. The boundary conditions can be observed in Fig.1.
<table>
<thead>
<tr>
<th>parameters</th>
<th>symbols</th>
<th>full scale model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scale factor</td>
<td>$\gamma$</td>
<td>57.5</td>
</tr>
<tr>
<td>Length between perpendiculars</td>
<td>$L_{PP}$ [$m$]</td>
<td>230</td>
</tr>
<tr>
<td>Length of waterline</td>
<td>$L_{WL}$ [$m$]</td>
<td>232.5</td>
</tr>
<tr>
<td>Maximum beam of waterline</td>
<td>$B_{WL}$ [$m$]</td>
<td>32.2</td>
</tr>
<tr>
<td>Draft</td>
<td>$T_0$ [$m$]</td>
<td>10.8</td>
</tr>
<tr>
<td>Displacement volume</td>
<td>$\nabla$ [$m^3$]</td>
<td>52,030</td>
</tr>
<tr>
<td>Block coefficient</td>
<td>$C_B$</td>
<td>0.6505</td>
</tr>
<tr>
<td>Longitudinal center of buoyancy</td>
<td>$LCB$ (%$L_{PP}$)</td>
<td>-1.48</td>
</tr>
<tr>
<td>Wetted surface area without rudder</td>
<td>$S_W$ [$m^2$]</td>
<td>9424</td>
</tr>
<tr>
<td>Moment of Inertia</td>
<td>$K_{xx}/B$</td>
<td>0.4</td>
</tr>
<tr>
<td></td>
<td>$K_{yy}/B$</td>
<td>0.25</td>
</tr>
<tr>
<td></td>
<td>$K_{zz}/B$</td>
<td>0.25</td>
</tr>
</tbody>
</table>

Table 1: Physical and geometrical parameters of the KCS model

3 COMPUTATIONAL TESTS

The computations are carried out for the KCS, whose parameters are shown in Tab.1. The computational results at $Fr = 0.26$ are compared against the experimental measurements from the Tokyo 2005 CFD workshop [2]. The results show good agreements. However some deviations can be observed behind the ship, demonstrating that the mesh there is still not fine enough.

![Mesh generation and boundary conditions.](image.png)
4 RESULTS AND DISCUSSIONS

Simulations with different drafts are conducted to investigate the influence of drafts. The range of drafts in this study is $T = T_0, 0.9T_0, ..., 0.5T_0$. The ‘interDyMfoam’ solver in the OpenFOAM is used here, which allows the mesh morphing. The free trim and sinkage of the ship can then be realized using this solver. In Fig.5, the mesh morphing can be clearly observed.

In Fig.6, it can be observed that with a smaller draft, the ship takes more time to become stable. Thereby large draft contributes to the ship stability.

In Fig.7(a)-(d), it can be clearly seen that the bulbous bow can modify the flow pattern, which can reduce the drag and increase the ship speed. With the decrease of the draft, the bulbous bow is gradually exposed into the air. Thus its effect of manipulating the flow and reducing the drag will be lost.

5 CONCLUSIONS

- The flow fields of the KCS ship were simulated and showed good agreements with the experimental data.

- Small draft is found to increase the instability of the ship and weaken the effect of the bulbous bow.
Figure 3: Wave profile on the hull surface ($Fr = 0.26$). 'o', experimental results; '-', computational results.

Figure 4: Wave elevation contours ($Fr = 0.26$): (a) experimental results; (b) computational results.
Figure 5: Mesh morphing during the simulations.

Figure 6: Comparison of unsteady simulations with different drafts ($Fr = 0.227$). $T_0$ is the original draft in Tab.1.
Figure 7: Free surfaces with different drafts ($Fr = 0.227$): (a) $T = T_0$; (b) $T = 0.9T_0$; (c) $T = 0.8T_0$; (d) $T = 0.7T_0$; (e) $T = 0.6T_0$; (f) $T = 0.5T_0$; (g) $T = 0.4T_0$. 
REFERENCES


INVESTIGATION OF THE CONFINEMENT EFFECT ON HYDRODYNAMIC DERIVATIVES OF A 135M INLAND CONTAINERSHIP

RAZGALLAH I. *, KAIDI S. * † AND SMAOUI H. *†

*Centre de Recherche Royallieu, Laboratoire Roberval
Université de Technologie de Compiègne
CS 60319,60203 Compiègne Cedex, France
e-mail: intissar.rezgallah@utc.fr, web page: http://www.utc.fr

†Centre d’études et d’expertise sur les risques, l’environnement, la mobilité et l’aménagement (CEREMA-DTeCEMF)
134, rue de Beauvais -CS 60039, 60280 Margny Lès Compiègne, France

Key words: Hydrodynamic derivatives, PMM, RANSE, maneuverability, confined waters

Abstract. The estimation of hydrodynamic interaction forces occurring when a ship is navigating in confined waterways plays an essential role in inland maneuverability modeling. The forces include the bottom effect which significantly affects the current loads and the flow around the hull. Therefore, this article deals with the evaluation of ship-bottom interaction force. A Three Degrees of Freedom model (3 DOF) is used in these simulations to account for surge, sway and yaw motion of the studied containership. The model’s hydrodynamic derivatives are determined based on CFD generated hydrodynamic forces. Using a RANSE solver, computerized Planar Motion Mechanism (PMM) tests are simulated at three values of depth to draft ratios. Based on the identified hydrodynamic coefficients, a comparative study is carried out with the semi-empirical formulations in published literature.

1 INTRODUCTION

Doing maneuverability prediction with a variety of scenarios is of great importance to guarantee the safety of a ship. According to the PIANC recommendations [1], the ship-bottom interaction depends on the ratio \((h/T)\) of water depth \((h)\) to the ship’s drought \((T)\). When this ratio is below three (depth restriction), the bottom effect becomes significant, thus causing a greater change in water velocity and pressure distribution around the hull which severely affects the ship’s maneuverability. Several studies have discussed the underlying physics and have attempted to model these effects [3, 4]. However, most of the investigations carried out in this field rely on experimental or semi-empirical methods to describe the interaction force between the ship and the channel bed. In particular, captive model testing, is known to be
the most widely accepted approach, as it offers access to a proper mathematical model of the ship’s motion. Due to progress in modern CFD techniques, simulation by a fully CFD approach has proven its reliability in deep water\cite{5, 6, 7}. In fact, instead of carrying out the Planar Motion Mechanism (PMM) tests in a towing tank, the tests are reproduced numerically. This method, not only, allows avoiding the expensive costs of experimentation, yet helps to considerably reduce the scale effect caused by the incapacity to achieve Froude and Reynolds similarities simultaneously, and the method gives access to detailed flow field around the hull.

In spite of less numerically based publications found on simulations in confined waters, CFD methods are gaining importance in the field of maneuverability. Most recently, the International Conference on Ship Maneuvering in Shallow and Confined Water held in Hamburg in 2016, tried to fill this gap. Among the existing computerized fluid dynamic studies on the water depth effect, presented during this conference, He et al.\cite{8}, ran a series of systematic computations of pure sway motion tests at \((h/T)\) ratios of 10, 3, 1.5 and 1.2, at a constant speed. He provides a qualitative insight into the bottom effect on the hydrodynamic forces. Furukawa et al. \cite{9}, focused on the longitudinal components of hydrodynamic derivatives for eighteen geometries of ship hulls deduced from captive model testing in shallow waters.

The aim of this work is to expand on the above mentioned studies to estimate the hydrodynamic coefficients of a 3DOF mathematical model in different channel configurations (different values of \(h/T\)) using a RANSE solver to numerically reproduce the PMM tests of a 1:25 scaled modeled containership hull. Note that in addition of the dynamic tests, our PMM tests include also the static drift and speed variation tests.

To properly apprehend the interaction between the ship’s hull and the bed of the canal, the hydrodynamic forces acting on the hull are determined numerically for various values of the ship’s speed. The numerical simulations are repeated for a number of canal configurations. The reliability of the numerical results are confirmed through the validation of standard tests in deep water.

2 PMM TESTS

The PMM tests are carried out on a bare hull model of a 135m inland containership scaled to 1:25. The hull characteristics are given in table 1. The ship’s geometry is symmetric to its midship. The tests include static tests (static drift and speed variation) and other dynamic tests (pure sway, pure yaw and yaw and drift). In static drift simulations, the drift angle was varied between zero and fifteen with a step of three degrees whereas in dynamic tests, a single run method was adopted in which the dimensionless frequency was set to 7.753, the dimensionless maximum sway amplitude was 0.03, the dimensionless maximum yaw amplitude was set to 0.0196 and the drift angle varies according to the dynamic test between zero and three degrees. These values are in accordance to the I.T.T.C recomendations concerning the restriction of oscillation frequency presented in “Recommended Procedures and Guidelines : Captive Model Test Procedure ”\cite{10}. An overview of the dynamic tests is given in figure 1. The maximum full scaled ship forward moving speed is no higher than 19 Km/h for all tests corresponding to a Froude number between 0.137 and 0.067.
Table 1: Geometric parameters of ship hull

<table>
<thead>
<tr>
<th></th>
<th>Length (LPP)</th>
<th>Beam (B)</th>
<th>Draft (T)</th>
<th>Block coefficient (CB)</th>
<th>Wetted surface (WS)</th>
<th>Cross area of ship (CS)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Real model</td>
<td>135 m</td>
<td>11.4 m</td>
<td>2.5 m</td>
<td>0.899</td>
<td>2104.8 $m^2$</td>
<td>34.114 $m^2$</td>
</tr>
<tr>
<td>Scaled model (1:25)</td>
<td>5.4 m</td>
<td>0.456 m</td>
<td>0.1 m</td>
<td>0.899</td>
<td>3.367 $m^2$</td>
<td>0.0545 $m^2$</td>
</tr>
</tbody>
</table>

3 NUMERICAL SIMULATIONS

The steady and unsteady averaged Navier-Stokes equations (URANSE) are solved using a finite volume solver under ANsys FLUENT to simulate respectively the dynamic tests. For system closure, the $k - \omega$ SST (shear stress transport) model is chosen for its ability to insure a good description of the entire boundary layer. Computations were performed on 1:25 scaled model to reduce the computational time. The geometry of the domain is set such that it extends $1L_{pp}$ from the bow and $2L_{pp}$ to the aft of the ship, and is distanced with $1L_{pp}$ for each from port and starboard sides in respect to the 2011 ITTC recommendations [11]. The boundary conditions were set as follows. At the inlet, velocity value was imposed. For the outlet, a pressure was adopted to prevent back-flow from occurring. For top and sides, a symmetry condition was applied and the boundary condition around the ship’s hull is set to a no slip wall. Depending on the simulated PMM test, the cylinder in which the ship is placed can be subject to periodic rotatory motion. In such cases, its use helps to conserve an appropriate mesh near the boundary.
layer. The bottom is fixed as a wall moving with the free stream speed. More details are found in figure 2. Computations use an unstructured grid. In order to mesh the complex geometry in the proximity of the hull, tetrahedral cells were used, whereas the rest of the fluid domain grid is generated using a multi-block hexahedral mesh. Different grid sizes have been tested for static drift test and based on the obtained results, a grid of about 1211809 cells was adopted, whereas in dynamic tests, the grid size was roughly 2436701 cells. The simulations were carried out for a ship navigating in deep water (h/T=5), in medium deep water (h/T=2) and in shallow water (h/T=1.2). The numerical method was validated for different channel configurations by comparison to towing tank measurements. The validation tests are reported in [12].

4 MATHEMATICAL MODEL OF SHIP MOTION

The ship’s trajectory simulation is based on the definition of two coordinate systems. The motion is described using a body-fixed coordinate system related to the ship and which moves relatively to the inertial reference frame \((O, x_e, y_e, z_e)\). This local frame of motion denoted by \((x_G, x, y, z)\), has its origin set at the center of gravity of the ship (figure 3) with the external forces being applied on the surrounding fluid balancing the inertial forces acting on the ship. The metacentric height is high enough as to allow that the roll is neglected. As for inland navigation, the ship rarely encounters severe wether conditions, the mathematical model is limited to to surge, sway and yaw.

The equations of motion are written in equation 1 by:

\[
\begin{align*}
(m + m_x) \ddot{u} - (m + m_y) \dot{v} \dot{r} &= X \\
(m + m_y) \ddot{v} + (m + m_x) \dot{u} \dot{r} &= Y \\
(I_{zz} + J_{zz}) \dot{r} &= N
\end{align*}
\]

\(X\) and \(Y\) are the external forces applied to the ship and projected respectively on the \(x\)- and \(y\)-axis and \(N\) is the moment of yaw expressing the rotational forces around the \(z\)-axis. On the left hand side of equation 1, \(m\) refers to the ship’s mass, \(\ddot{u}, \ddot{v}\) and \(\dot{r}\) are respectively the cinematic
and angular accelerations, while \( r \) is the yaw angular velocity. \( I_{zz} \) is the moment of inertia along the \( z \)-axis. \( m_x, m_y \) and \( J_{zz} \) are terms due to added mass and added inertia. The external forces and yaw moment gather all the forces acting on the propeller, the rudder, the hull and interaction effects due to confinement. The forces are written in an dimensionless form using the prime-system II. The dimensionless forces are:

\[
X' = \frac{X}{0.5\rho U_0 L_{pp} T}, \quad Y' = \frac{Y}{0.5\rho U_0 L_{pp} T}, \quad N' = \frac{N}{0.5\rho U_0 L_{pp} T}.
\]

The forces are supposed to be quasi-steady and depend mainly on \( u, v, r, \dot{u}, \dot{v}, \dot{r} \) and \( \delta \) in calm water and can be described using Abkowitz’s model[13] modified by Strom-Tejsen[14].

\[
X' = X + X_u \dot{u}' + X_u \Delta u' + X_{uu} \Delta u'^2 + X_{uuu} \Delta u'^3 + X_{vuv} \dot{v}' + X_{ruu} \dot{r}' + X_{ruv} \dot{v}' \Delta u' + X_{r} \dot{r}' \Delta u' + X_{\delta\delta} \Delta u' + X_{\delta\delta}\delta
\]

\[
Y' = Y + Y_u \Delta u' + Y_{uu} \Delta u'^2 + Y_{uuu} \Delta u'^3 + Y_{vuv} \dot{v}' + Y_{ruu} \dot{r}' + Y_{ruv} \dot{v}' \Delta u' + Y_{r} \dot{r}' \Delta u' + Y_{\delta\delta} \Delta u' + Y_{\delta\delta}\delta\delta
\]

\[
N' = N_u \Delta u' + N_{uu} \Delta u'^2 + N_{uuu} \Delta u'^3 + N_{\delta\delta} \Delta u' + N_{\delta\delta}\delta \Delta u' + N_{\delta\delta}\delta \delta \Delta u' + N_{\delta\delta}\delta \delta \delta \Delta u' + N_{\delta\delta}\delta \delta \delta \delta \Delta u' + N_{\delta\delta}\delta \delta \delta \delta \delta \Delta u' + N_{\delta\delta}\delta \delta \delta \delta \delta \delta \Delta u' + N_{\delta\delta}\delta \delta \delta \delta \delta \delta \delta \Delta u' + N_{\delta\delta}\delta \delta \delta \delta \delta \delta \delta \delta \Delta u' + N_{\delta\delta}\delta \delta \delta \delta \delta \delta \delta \delta \delta \Delta u'
\]

Where \( u' \) and \( v' \) are dimensionless linear velocities in \( x- \) and \( y \)-direction, respectively, and \( r' \) is the non-dimensional angular velocity in the \( z \)-direction and \( \Delta u \) is the disturbance in surge velocity. \( X' \) is the reference steady state value of \( X' \) and tends towards zero when the ship is advancing at a constant speed (similar to speed variation test configuration). The first-order hydrodynamic derivatives of forces and moment are devided into velocity derivatives denoted by \( X_u, Y_u, N_u, Y_r, N_r \) and acceleration derivatives \( X_{\ddot{u}}, Y_{\ddot{u}}, N_{\ddot{u}}, Y_{\ddot{r}}, N_{\ddot{r}} \). The second order derivatives
are divided into uncoupled hydrodynamic coefficients mainly $X_{uu}, X_{uv}, X_{rr}$ and cross-coupled hydrodynamic derivatives $X_{uv}$. In a similar way, the third order derivatives are divided into uncoupled $X_{uuu}, Y_{vvv}, Y_{rrr}, N_{vvv}, N_{rrr}$ and cross-coupled coefficients regrouping $Y_{urr}, Y_{rue}, N_{urr}, N_{rve}$.

Since the simulations are carried out on the bare hull, the terms $\delta$ related to the rudder can be omitted.

5 IDENTIFICATION OF HYDRODYNAMIC DERIVATIVES

5.1 Tests results

When UKC or $(h/T)$ decreases, the shallow water effect increases drastically leading to an increase in wave-making resistance. This results in a huge wave resistance for small UKC numbers and the pressure resistance becomes significant, which explains the increase in the $X'$ total force observed during the simulation tests. In addition to this, it is expected that the resistance force in pure sway tests is smaller than that in the yaw tests. Since during the latter, the ship is constantly changing its heading in order to follow the path which results in a variation of the projected area and thus, too, the forces. Also, the reduction of water depth causes a strong increase of transverse forces and moments as the ship turns. This can be seen on figures 4, 5 and 6, where the time history of pure sway, pure yaw, and yaw and drift tests are shown. It is also possible to verify the symmetry of the pure sway and pure yaw tests with respect to the direction of motion. The symmetry is completely lost in yaw and drift test due to the asymmetric nature of the test.

**Figure 4:** Time history of dimensionless force/moment for dynamic simulation in pure sway mode

**Figure 5:** Time history of dimensionless force/moment for dynamic simulation in pure yaw mode
5.2 Identified coefficients

The sway velocity derivatives can be identified using static drift test by fitting the plot of the longitudinal force to the drift angle using a second order polynomial via:

\[ X' = X_{vv} \beta^2 + X_* \]  \hspace{1cm} (4)

The sway force and yaw moment are fitted using a third order polynomial according to the following

\[ (Y', N') = (Y_{vvv}, N_{vvv}) \beta^3 + (Y_v, N_v) \beta \]  \hspace{1cm} (5)

In addition to these coefficients, the resistance derivatives are extracted from static speed variation test. For this test, the ship is moving forward at a constant speed with the drift angle set to zero. The surge force is plotted against the advancing speed (see figure 7), and the curve is fitted afterwards using a third order polynomial and the dimensionless \( X_{uuu}, X_{uu} \) and \( X_u \) are determined by dividing the polynomial parameters by \( 0.5 \rho LT \).

\[ X_{uuu}, X_{uu}, X_u \]

The sway acceleration derivatives can only be deduced from pure sway tests. In order to identify the derivatives from pure sway test, the forces and yaw moment time history presented in figure 4 are fitted using a second (for X force) and third order Fourier series as explained.
bellow.

\[
X' = X + \frac{1}{2}X_{ve} v_{max}^' + \frac{1}{2}X_{ve} v_{max}^2 \cos(2\omega't')
\]
\[
Y' = -(Y_v v_{max}^' + \frac{3}{4} Y_{vv} v_{max}^3) \cos(\omega't') + Y_v \omega' v_{max}^' \sin(\omega't') - \frac{1}{4} Y_{ve} v_{max}^3
\]
\[
N' = -(N_v v_{max}^' + \frac{3}{4} N_{vv} v_{max}^3) \cos(\omega't') + N_v \omega' v_{max}^' \sin(\omega't') - \frac{1}{4} N_{ve} v_{max}^3
\]

A good agreement is found in general between the repeating derivatives of pure sway and static drift with a ratio of the value of respectively \(Y_v\) and \(N_v\) in static drift to its value obtained through pure sway test varying roughly between 0.85 and 1.23, and average value of the same ratio of \(Y_{vv}\) and \(N_{vv}\) no bigger than to 1.09. Yaw rate derivatives coefficients are determined, in this study, through simulations of pure yaw tests. Similarly, for pure sway test, the derivatives are identified using Fourier series fitting for \(t\), the following expressions for each force and moment:

\[
X' = X + \frac{1}{2}X_{rr} r_{max}^' + \frac{1}{2}X_{rr} r_{max}^2 \cos(2\omega't')
\]
\[
Y' = (Y_r r_{max}^' + \frac{3}{4} Y_{rr} r_{max}^3) \sin(\omega't') + Y_r \omega' r_{max}^' \cos(\omega't') - \frac{1}{4} Y_{rr} r_{max}^3
\]
\[
N' = (N_r r_{max}^' + \frac{3}{4} N_{rr} r_{max}^3) \sin(\omega't') + N_r \omega' r_{max}^' \cos(\omega't') - \frac{1}{4} N_{rr} r_{max}^3
\]

Finally, coupled derivatives are identified using yaw and drift tests by Fourrier fitting in respect to the following form:

\[
X' = X + \frac{1}{2}X_{rr} r_{max}^' + X_{rr} r_{max}^2 + X_{ve} v' + X_{ve} v_{max}^' \sin(\omega't') - \frac{1}{2}X_{rr} r_{max}^2 \cos(2\omega't')
\]
\[
Y' = Y_v v' + \frac{1}{2} Y_{rr} r_{max}^2 + Y_{fr} v_{fr}^2 + \left( Y_r r_{max}^' + \frac{3}{4} Y_{rr} r_{max}^3 + Y_{ve} v_{max}^' v^2 \right) \sin(\omega't')
\]
\[
+ Y_r \omega' r_{max}^' \cos(\omega't') - \frac{1}{4} Y_{rr} r_{max}^3 \cos(2\omega't') - \frac{1}{4} Y_{rr} r_{max}^3 \cos(3\omega't')
\]
\[
N' = N_v v' + \frac{1}{2} N_{rr} r_{max}^2 + N_{ve} v_{ve}^2 + \left( N_r r_{max}^' + \frac{3}{4} N_{rr} r_{max}^3 + N_{ve} r_{max}^' v^2 \right) \sin(\omega't')
\]
\[
+ N_r \omega' r_{max}^' \cos(\omega't') - \frac{1}{4} N_{rr} r_{max}^3 \cos(2\omega't') - \frac{1}{4} N_{rr} r_{max}^3 \cos(3\omega't')
\]

The identified coefficients are presented in table 2.

6 COMPARATIVE STUDY

6.1 Comparison of computed derivatives with semi-empirical correlations in deep water

As mentioned earlier, the sway velocity derivatives are determined through static drift and pure sway tests. The static drift test is known to yield more reliable results, since the dynamic tests are frequency dependent. A comparison of the identified coefficients to the existing semi-empirical expressions developed by Inoue, Clarke, Norrbin, Wagner Smitt, Jones in deep water
show good agreement with the coefficients determined numerically (see table 3). Based on the aspect ratio theory, the correlations according to Jones, under estimates all the hydrodynamic coefficients presented in the table except for $Y_z$. The rest of the correlations presented are derived using empirical methods. It has been noticed that Clarke correlations tend to overestimate the derivatives, while the empirical relations proposed by Wagner Smitt tends to show the inverse behaviour. Finally, the best results are obtained using the correlations of Norrbin and Inoue (The maximum ratio of the numerically obtained derivatives to those calculated using the correlations equal respectively to 2.3 and 2.4 and the minimum ratio is 0.63 and 0.7). Some discrepancies are observed for $N_v$. This might be due to the inaccuracy of the semi-empirical methods when it is used for other hull shapes than the ones they were developed for. It is worth noticing that the most important derivatives are $X_{uu}$, $Y_v$, $N_v$, $Y_r$, $N_r$, $Y_v$. The identified hydrodynamic derivatives will be implemented in a further work into the chosen mathematical model which will be used to simulate standard tests (giration, zig zag, crash-stop..) and compared to in-situ tests results for validation.

Table 3: Comparison of coefficients in deep water($h/T = 5$)

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$Y_z$</td>
<td>-0.0581</td>
<td>-0.0924</td>
<td>-0.1043</td>
<td>-0.1535</td>
<td>-0.1644</td>
<td>-0.1158</td>
<td>-0.1227</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$N_v$</td>
<td>-0.0291</td>
<td>-0.0360</td>
<td>0.0346</td>
<td>-0.0316</td>
<td>-0.0370</td>
<td>-0.0811</td>
<td>-0.0986</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$Y_r$</td>
<td>0.0291</td>
<td>-0.0186</td>
<td>-0.0402</td>
<td>0.0395</td>
<td>-0.0290</td>
<td>-0.0254</td>
<td>-</td>
<td>-0.0153</td>
<td>-</td>
</tr>
<tr>
<td>$N_r$</td>
<td>-0.0145</td>
<td>-0.0122</td>
<td>-0.0136</td>
<td>-0.0221</td>
<td>-0.0186</td>
<td>-1</td>
<td>-0.1317</td>
<td>-0.1381</td>
<td>-</td>
</tr>
<tr>
<td>$Y_{err}$</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-0.3950</td>
<td>-5.663000</td>
<td>-15.776800</td>
<td>-</td>
</tr>
<tr>
<td>$N_{err}$</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-0.0490</td>
<td>-0.060800</td>
<td>0.182700</td>
<td>-</td>
</tr>
<tr>
<td>$N_{ave}$</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-0.7399</td>
<td>-1.890100</td>
<td>-3.973400</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 2: Numerically identified derivatives

<table>
<thead>
<tr>
<th>Coefficients</th>
<th>Pure sway</th>
<th>Pure yaw</th>
<th>yaw and drift</th>
</tr>
</thead>
<tbody>
<tr>
<td>$X_{uu}$</td>
<td>0.0056</td>
<td>0.00647</td>
<td>0.0652</td>
</tr>
<tr>
<td>$X_{uv}$</td>
<td>-0.0516</td>
<td>-0.0567</td>
<td>-0.0626</td>
</tr>
<tr>
<td>$X_{uv}$</td>
<td>-0.0028</td>
<td>-0.0031</td>
<td>-0.0035</td>
</tr>
<tr>
<td>$X_{vv}$</td>
<td>-0.0282</td>
<td>-0.0318</td>
<td>-0.0388</td>
</tr>
<tr>
<td>$X_{rv}$</td>
<td>-0.0743</td>
<td>-0.1093</td>
<td>-0.1688</td>
</tr>
<tr>
<td>$Y_{uv}$</td>
<td>-0.1158</td>
<td>-0.2592</td>
<td>-1.5167</td>
</tr>
<tr>
<td>$Y_{rv}$</td>
<td>-1.5567</td>
<td>-3.5727</td>
<td>-6.7825</td>
</tr>
<tr>
<td>$Y_{vv}$</td>
<td>-0.0811</td>
<td>-0.1435</td>
<td>-0.4993</td>
</tr>
<tr>
<td>$N_{uv}$</td>
<td>0.0956</td>
<td>0.0756</td>
<td>0.5200</td>
</tr>
<tr>
<td>$N_{uv}$</td>
<td>-0.1158</td>
<td>-0.1679</td>
<td>-0.4806</td>
</tr>
<tr>
<td>$N_{rv}$</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$N_{rv}$</td>
<td>-0.0028</td>
<td>-0.0031</td>
<td>-0.0035</td>
</tr>
<tr>
<td>$N_{rv}$</td>
<td>-0.0282</td>
<td>-0.0318</td>
<td>-0.0388</td>
</tr>
<tr>
<td>$N_{rv}$</td>
<td>-0.0743</td>
<td>-0.1093</td>
<td>-0.1688</td>
</tr>
<tr>
<td>$Y_{rv}$</td>
<td>-0.1158</td>
<td>-0.2592</td>
<td>-1.5167</td>
</tr>
<tr>
<td>$Y_{rv}$</td>
<td>-1.5567</td>
<td>-3.5727</td>
<td>-6.7825</td>
</tr>
<tr>
<td>$Y_{rv}$</td>
<td>-0.0811</td>
<td>-0.1435</td>
<td>-0.4993</td>
</tr>
</tbody>
</table>
6.2 Comparison of computed derivatives with semi-empirical correlations in shallow water conditions

In shallow water, it was possible to compare the coefficients obtained numerically to those provided by Ankudinov et al. and Clarke et al. These semi-empirical correlations are based on using a correction factor of deep to shallow water coefficients. The correlation developed by Clarke is valid for $1.25 < h/T < \infty$. It is noticed from table 4 that Clarke’s correlations yield inaccurate results for $h/T = 1.2$. Ankudinov’s relations are adapted for smaller under keel clearance values $1.085 < h/T < 5$ and a block coefficient smaller than 0.85. Although the simulated ship has a slightly bigger block coefficient, Ankudinov’s correlation gives a fairly good approximation of the derivatives.

### Table 4: Comparison of some hydrodynamic coefficients in shallow water

<table>
<thead>
<tr>
<th>Coefficients</th>
<th>Clarke et al. [18]</th>
<th>Ankudinov et al. [20]</th>
<th>Numerical simulations</th>
</tr>
</thead>
<tbody>
<tr>
<td>$N_{w}$</td>
<td>$-0.1747$</td>
<td>$-0.0591$</td>
<td>$-0.1699$</td>
</tr>
<tr>
<td>$N_{v}$</td>
<td>$0.0038$</td>
<td>$0.0151$</td>
<td>$0.0016$</td>
</tr>
<tr>
<td>$N_{x}$</td>
<td>$-0.1950$</td>
<td>$-0.0748$</td>
<td>$-0.1688$</td>
</tr>
<tr>
<td>$N_{y}$</td>
<td>$-0.8499$</td>
<td>$-0.5079$</td>
<td>$-0.4806$</td>
</tr>
<tr>
<td>$N_{z}$</td>
<td>$-3.1197$</td>
<td>$-0.9587$</td>
<td>$-3.5771$</td>
</tr>
<tr>
<td>$N_{r}$</td>
<td>$-0.4326$</td>
<td>$-0.1329$</td>
<td>$0.0185$</td>
</tr>
<tr>
<td>$N_{r}$</td>
<td>$-1.4847$</td>
<td>$-1.1521$</td>
<td>$-0.3897$</td>
</tr>
<tr>
<td>$N_{y}$</td>
<td>$-1.2644$</td>
<td>$-0.5556$</td>
<td>$-0.1542$</td>
</tr>
<tr>
<td>$N_{z}$</td>
<td>$0.1864$</td>
<td>$0.0432$</td>
<td>$0.0790$</td>
</tr>
<tr>
<td>$N_{r}$</td>
<td>$-0.7240$</td>
<td>$-0.1679$</td>
<td>$-0.5961$</td>
</tr>
<tr>
<td>$N_{r}$</td>
<td>$-0.0875$</td>
<td>$-0.0233$</td>
<td>$-0.0547$</td>
</tr>
<tr>
<td>$N_{x}$</td>
<td>$0.1826$</td>
<td>$0.0618$</td>
<td>$0.1827$</td>
</tr>
<tr>
<td>$N_{y}$</td>
<td>$-0.6323$</td>
<td>$-0.1801$</td>
<td>$-0.6327$</td>
</tr>
<tr>
<td>$N_{z}$</td>
<td>$-0.1801$</td>
<td>$-0.6327$</td>
<td>$-0.1801$</td>
</tr>
</tbody>
</table>

7 CONCLUSIONS

Using RANS and URANS models, a series of numerical simulations were carried out in this study to identify the hydrodynamic derivatives of a 135 m inland containership for three values of depth to draft ratio. The latter were compared to values of the derivatives obtained semi-empirical correlations. It has been observed that most of the semi-empirical formulas are defined for a certain range of $h/T$ ratios. This study shows that some correlations yield a good estimation of certain coefficients and some don’t.

- In deep water, the correlation developed by Norrbin fits best $Y_{v}$ (with $Y_{v,\text{Num}}/Y_{v,\text{Norrbin}} = 1.2$) and $N_{r}$ (with $N_{r,\text{Num}}/N_{r,\text{Norrbin}} = 1.12$) Whereas the correlation by Inoue is best suited for the estimation of $Y_{r}$ (with $Y_{r,\text{Num}}/Y_{r,\text{Inoue}} = 0.87$). Jones’s semi-empirical relation also shows acceptable results for $N_{r}$ estimation (with $N_{r,\text{Num}}/N_{r,Jones} = 1.05$).

- The earlier correlations showed a great inaccuracy in the estimation of $N_{v}$ coefficient. The biggest discrepancy is observed when comparing numerical results with Jones’s correlation ($N_{v,\text{Num}}/N_{v,Jones} = 2.7$) and the smallest when comparing with Inoue correlation ($N_{v,\text{Num}}/N_{v,\text{Inoue}} = 2.19$).
Since the semi-empirical correlations allow only to identify first and second order derivatives, no correlations could be found in the literature for third order derivatives ($Y_{vvv}$, $N_{vvv}$, $Y_{rrr}$ and $N_{rrr}$).

In shallow water, the approximation of hydrodynamic derivatives using deep water coefficients and $(h/T) - 1$ is recommended for the development of empirical expressions to the remaining hydrodynamic coefficients.

The CFD approach adopted in this article, has proven its reliability and can be used for further work. The next step would be to use the developed mathematical model to simulate standard tests.

REFERENCES


NUMERICAL ASSESSMENT OF INTERFERENCE RESISTANCE FOR A SERIES 60 CATAMARAN

ANDREA FARKAS*, NASTIA DEGIULI* AND IVANA MARTIĆ*

* Faculty of Mechanical Engineering and Naval Architecture (FMENA)
University of Zagreb
Ivana Lučića 5, 10000 Zagreb, Croatia
e-mail: andrea.farkas@fsb.hr, nastia.degiuli@fsb.hr, ivana.martic@fsb.hr, webpage: www.fsb.hr

Key words: Computational Fluid Dynamics (CFD), Volume of Fluid (VOF), k-ε turbulence model, interference resistance

Abstract. An important consideration in the catamaran design is the distance between the hulls. Arrangement of the hulls in catamaran configuration can have strong influence on the wave making resistance and thus on the total resistance of a catamaran. The hydrodynamic interaction between hulls becomes significant when spacing between hulls is sufficiently small. In this paper, numerical simulations of viscous flow around monohull and catamaran model are performed utilizing commercial software package STAR-CCM+, in order to investigate the influence of spacing between hulls on the interference resistance. A mathematical model based on Reynolds Averaged Navier-Stokes (RANS) equations, k-ε turbulence model and Volume of Fluid (VOF) method for describing the motion of two-phase media are briefly described. Numerical simulations are performed for Series 60 monohull and two catamaran configurations with $C_{f}=0.6$ for different values of Froude number. Results of performed numerical simulations are compared with experimental results available in the literature and satisfactory agreement has been achieved. It has been shown that CFD is a very useful tool in preliminary catamaran design.

1 INTRODUCTION

Catamaran configurations, as well as the other multihull configurations, have attracted attention because of their exquisite performance regarding the speed, safety, resistance, maneuverability and transversal stability. Multihulls have better technical characteristics than monohulls [1] and therefore a significant increase in demand for them can be noticed in the civil, recreational and military field. Even though numerous theoretical, numerical and experimental investigation concerning multihull vessels have been made recently [2], catamaran resistance is still very unpredictable [3]. Due to hydrodynamic interaction between hulls, spacing between them is one of the most important parameters in catamaran design and it must be taken into account at the design stage [4]. The wave systems of each hull usually strongly interfere, causing either favourable or unfavourable effects [5]. As a result of the interference resistance, catamaran resistance is not just a double resistance of the monohull [6]. Because of the significance of the spacing between hulls in catamaran configurations, many authors have investigated the relation between spacing between hulls and the
interference resistance.

Broglia et al. [7] carried out towing tank tests for Delft 372 catamaran and concluded that the interference effects are dominant for small spacing between hulls and at intermediate values of Froude number ($Fn$). At lower $Fn$ wave elevations are too small to produce significant effect on the total resistance of a catamaran and at higher $Fn$ the individual wave systems of each hull are very diverging and thus superposition between them is considerably reduced. Therefore, a catamaran starts to behave as a combination of almost non-interacting vessels. Trim and sinkage are found to be strongly related with the interference effects.

Due to advancements of computer science and numerical computation methods, the efficiency and accuracy of Computational Fluid Dynamics (CFD) methods are greatly improved. Consequently, the combination of model tests and CFD methods is becoming an optimum choice to analyze hydrodynamic characteristics of a catamaran [8]. Although catamarans have significant growth in popularity, numerical methods for the determination of their hydrodynamic characteristics are rather scarce and incomplete [4]. Haase et al. [9] developed a novel full-scale resistance prediction method for large medium-speed catamarans based on CFD. Their method is based on the assumption that accuracy of pressure drag is independent of Reynolds number ($Rn$). This method can successfully estimate full-scale resistance based on the simulations at full-scale $Rn$ without altering the linear dimensions, flow velocity or spatial resolution of the initial model mesh. Broglia et al. [10] have performed numerical simulations for both catamaran and monohull models in order to investigate interference effects and their dependence on $Rn$. These simulations have been performed for fixed models at the dynamic positions taken from the measurements. The analysis of the results showed that interference effects have weak dependence on $Rn$. In [11] Zaghi et al. have presented the results of extensive experimental and numerical studies, performed in order to investigate the interference effects and their dependence on the spacing between hulls. The interference resistance as well as the maximum of the total resistance coefficient are found to be higher for the narrower configurations. Maximum total resistance coefficient occurs at higher $Fn$ for the narrower configurations. Interference is considerably affected by the section shape of the demihull [12]. Yengejeh et al. [13] have performed various numerical simulations utilizing solver based on Reynolds Averaged Navier-Stokes (RANS) equations for asymmetric planing hulls at different trim angles, spacing between hulls and $Fn$. Authors have shown that catamaran configuration has significantly reduced wetted surface area than the corresponding monohull having the same displacement.

In this paper, numerical simulations of the viscous flow around monohull and catamaran models are performed utilizing commercial software package STAR-CCM+, in order to study the influence of spacing between hulls on the interference resistance. Mathematical and physical model used in numerical simulations are presented. Thereafter, numerical setup and implemented boundary conditions are given. Numerical simulations are performed for Series 60 monohull and catamaran with $C_b=0.6$ for $Fn$ values in the range from 0.3 to 0.5 and for two spacing between hulls 0.565 m and 0.971 m. Interference resistance is investigated through the interference factor ($IF$). Obtained $IF$ are compared with experimental results available in the literature and satisfactory agreement has been achieved.
2 GOVERNING EQUATIONS

Navier-Stokes equations and the continuity equation form a system of coupled, non-linear partial differential equations. Since, they are not solvable for flows around ship hulls, these equations are averaged. Time averaging of Navier-Stokes equations yields to RANS. RANS and averaged continuity equations are given as follows [14]:

\[
\frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial (\rho \bar{uu}_i + \rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \rho \frac{\partial \tau_{ij}}{\partial x_i}
\]

(1)

\[
\frac{\partial (\rho \bar{p})}{\partial x_i} = 0
\]

(2)

where \( \rho \) is the fluid density, \( \bar{u}_i \) is the averaged Cartesian components of the velocity vector, \( \rho \bar{u}_i \bar{u}_j \) is the Reynolds tensor stress and \( \bar{p} \) is the mean pressure. \( \tau_{ij} \) is the mean viscous stress tensor defined with following equation:

\[
\tau_{ij} = \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)
\]

(3)

where \( \mu \) is the dynamic viscosity.

For tracking and locating the free surface, Volume of Fluid (VOF) method is used. This method represents volume fraction occupied by some fluid inside an arbitrary closed volume. The volume fraction of water (\( \alpha_l \)) is determined according to continuity equation and for incompressible flow reads:

\[
\frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\alpha_l \bar{u}_i) = 0
\]

(4)

Physical properties of particular fluid depend on the presence of that fluid in the particular cell. If there are only two fluids present in the domain, fluid 1 and fluid 2, density is being calculated according to equation:

\[
\rho = \rho_1 (1 - \alpha_l) + \rho_2 \alpha_l
\]

(5)

where \( \rho_1 \) is the density of the fluid 1, \( \rho_2 \) is the density of the fluid 2 and \( \alpha_l \) is the volume fraction of the fluid 1. Other physical properties are calculated analogously according to equation (5).

Most commercial RANS solvers are based on Finite Volume Method (FVM). For description of turbulence effects on averaged flow, \( k-\varepsilon \) turbulence model is used together with wall functions. This is two equation model that solves transport equations for turbulent kinetic energy \( k \) and its dissipation rate \( \varepsilon \). In this paper, Realizable \( k-\varepsilon \) Two-Layer (RKE2L) turbulence model was used. This model generally gives at least as good, or even better results than Standard \( k-\varepsilon \) (SKE) turbulence model [15] which was proposed by Lauder and Spalding [16]. RKE2L contains a new transport equation for \( \varepsilon \) and critical coefficient of the model \( C_\mu \) is no longer constant as in the SKE, but it is instead expressed as a function of a mean flow and turbulence properties. Furthermore, RKE2L can work with low-\( Rn \) type meshes and with wall
function type meshes. This allows that $y^+$ parameter in the first cell can be either smaller than 1 or in the range $30 < y^+ < 1000$ [15].

3 COMPUTATIONAL MODEL

In this Section, description of computational model used within this research is presented. Firstly, the model used in numerical simulations is presented, then creation of the virtual towing tank is described and afterwards numerical setup is given.

3.1 Model geometry

Numerical simulations were performed for slightly modified S60 model compared to the one defined as benchmark for Tokyo 1994 CFD Workshop. This model is the same one as used in [5]. Modification is made because the hull geometry of benchmark model for Tokyo 1994 CFD Workshop had too many surface patches with not enough quality matching. Also, in order to cope with waves generated at higher $Fn$, vertical extension of the hull is made [5]. Body plan of the original S60 geometry [17] and the one used in [5] is shown in Figure 1. It can be seen that matching of these two geometries is satisfactory. The main particulars of the monohull model used in this paper are presented in Table 1. Two different catamaran configurations have been investigated. The alteration is based on the different spacing between hulls $s$, one catamaran configuration has $s=0.565$ m (C1) and another one has $s=0.971$ m (C2).

![Figure 1: The original S60 bodyplan [17] (black) and the modified bodyplane [5] (red)](image)

<table>
<thead>
<tr>
<th>Main particulars</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Length between perpendiculars ($L_{pp}$)</td>
<td>2.5 m</td>
</tr>
<tr>
<td>Beam ($B$)</td>
<td>0.333 m</td>
</tr>
<tr>
<td>Draft ($T$)</td>
<td>0.133 m</td>
</tr>
<tr>
<td>Wetted surface ($S$)</td>
<td>1.062 m²</td>
</tr>
<tr>
<td>Displacement ($\Delta$)</td>
<td>65.7 kg</td>
</tr>
<tr>
<td>Block coefficient ($C_B$)</td>
<td>0.6</td>
</tr>
</tbody>
</table>
3.2 Virtual towing tank

Virtual towing tank is made by creating a domain around the model hull. All domain boundaries are placed $2L_{pp}$ away from the model. Due to symmetry of the ship model, only half of computational domain is modeled. Thus, for monohull only half of the ship model is considered. For catamaran configuration, only demihull was considered. The centerline of demihull is placed at half of $s$ from the symmetry plane. Unstructured hexahedral mesh is made utilizing meshing tools within STAR-CCM+ as follows: Surface Remesher, Trimmer, Prism Layer Mesher and Automatic Surface Repair. All mesh parameters are defined as relative values of the cell base size, except in the case of Prism Layer Mesher, where prism layer thickness is set as an absolute value in order to keep the same value of $y^+$ parameter in the first cell next to the wall. This value is held above 30 in order to ensure that the near-wall cell lies within the logarithmic region of the boundary layer. Prism layer is made with six cells. Two grids with different number of cells are made, coarse (G1) and fine (G2). Coarse grids for both monohull and catamaran configurations had around half of million cells, while fine grid for monohull had around 1.9 million cells and for catamaran configurations around 2.1 million cells. The structure of G2 for monohull and C2 can be seen in Figure 2. Calculated $y^+$ at the first cell next to the wall, for monohull with G2 and $Fn=0.45$ is shown in Figure 3.

Figure 2: Fine grid G2 for monohull (upper) and for C2 (lower)
Figure 3: Calculated $y^+$ distribution for monohull in the first cell next to the wall for $Fn=0.45$ with fine grid G2

3.3 Numerical setup

The governing equations described in the Section 2 are discretized using a cell based FVM. Temporal discretization is made using a first-order temporal scheme, also referred as Euler implicit. Convection terms in RANS were discretized with a second-order upwind scheme. Under relaxation factor for velocity is set to 0.7 and for pressure to 0.4. As said before, VOF method is used for modeling the free surface and High Resolution Interface Capturing Scheme (HRIC) is used to track sharp interfaces. Applied boundary conditions are shown in Figure 4.

Figure 4: Applied boundary conditions

Time step in every simulation is set to $T/200$, where $T$ is the ratio between $L_{pp}$ and velocity imposed at inlet boundary. Reflection of VOF waves is prevented by importing VOF wave damping at inlet, outlet and side boundary. Implemented approach in STAR-CCM+ is proposed by Choi and Yoon [15]. In this paper, VOF wave damping length is defined using $dvar$ function. This function is presented with the following equation:

$$dvar \approx L + L\cos^2\left(\frac{\pi L}{2 \cdot 10T}\right)$$

This function dampens almost entire area around the ship model at the beginning. As physical time passes, smaller part of the domain is being damped up till $10T$. From then on, half of the domain is damped until the end of simulation. Using the larger damping zone in the beginning of the simulation ensures faster convergence of the results.
4 RESULTS AND DISCUSSION

The total resistance of S60 monohull and two catamaran configurations C1 and C2 is calculated for four different values of \( Fn \) in the range from 0.3 to 0.5 and the obtained results are compared with experimental results published in [5]. Simulations were performed for fixed models and comparison of the obtained results using two different grid densities with experimental data is shown in Table 2. The obtained results are given as total resistance values, as well as relative deviations (\( RD \)) from experimental results. \( RD \) is calculated according to the following equation:

\[
RD = \frac{R_{CFD} - R_{EXP}}{R_{EXP}}
\]

where \( R_{CFD} \) is the total resistance obtained utilizing CFD and \( R_{EXP} \) is the total resistance obtained experimentally.

Table 2: Comparison between experimentally and numerically obtained values of the total resistance

<table>
<thead>
<tr>
<th>( Fn )</th>
<th>( R_{Tmh}^{E} )</th>
<th>( R_{TC1}^{E} )</th>
<th>( R_{TC2}^{E} )</th>
<th>( R_{Tmh}^{G1} )</th>
<th>( R_{TC1}^{G1} )</th>
<th>( R_{TC2}^{G1} )</th>
<th>( R_{Tmh}^{G2} )</th>
<th>( R_{TC1}^{G2} )</th>
<th>( R_{TC2}^{G2} )</th>
<th>( RD )</th>
<th>( RD )</th>
<th>( RD )</th>
<th>( RD )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.35</td>
<td>8.643</td>
<td>18.899</td>
<td>17.217</td>
<td>9.217</td>
<td>19.084</td>
<td>17.400</td>
<td>9.167</td>
<td>19.296</td>
<td>17.675</td>
<td>(+6.64)</td>
<td>(+0.98)</td>
<td>(+1.06)</td>
<td>(+2.10)</td>
</tr>
<tr>
<td>0.4</td>
<td>17.089</td>
<td>35.223</td>
<td>43.320</td>
<td>17.991</td>
<td>35.990</td>
<td>45.599</td>
<td>17.967</td>
<td>36.601</td>
<td>45.213</td>
<td>(+5.28)</td>
<td>(+2.18)</td>
<td>(+5.26)</td>
<td>(+3.91)</td>
</tr>
<tr>
<td>0.5</td>
<td>36.375</td>
<td>98.814</td>
<td>81.635</td>
<td>37.863</td>
<td>102.411</td>
<td>85.374</td>
<td>37.870</td>
<td>102.243</td>
<td>85.113</td>
<td>(+4.09)</td>
<td>(+3.64)</td>
<td>(+4.58)</td>
<td>(+3.47)</td>
</tr>
</tbody>
</table>

The results obtained with numerical simulations show satisfactory agreement with experimental results. As it can be seen from Table 2, numerical results overestimate experimental results for monohull and both catamaran configurations for all four values of \( Fn \). The overestimation is more significant for monohull for lower values of \( Fn \). The greatest relative deviation for monohull using the grid G2 is 10.73% for \( Fn=0.3 \). It should be mentioned that \( R_{EXP} \) values are experimentally obtained in kilograms (kg). This means that \( R_{EXP} \) value for \( Fn=0.3 \) amounts 0.5855 kg. For such a small value, even small mistake or uncertainty in experimental measurement can lead to relatively high overestimation. The greatest relative deviation for C1 using fine grid G2 is 3.91% and for C2 is 5.47%. The greatest relative deviation obtained using coarse grid G1 for monohull, C1 and C2 are 11.00%, 4.47% and 6.79% respectively.

Within this research, interference resistance of S60 catamaran is investigated through \( IF \). \( IF \) is defined as the ratio of the difference between the total resistance of the catamaran \( R_{cat} \) and twice the total resistance of the monohull \( R_{Tmh}^{E} \), and twice the total resistance of the monohull:

\[
IF = \frac{R_{cat} - 2 \times R_{Tmh}^{E}}{2 \times R_{Tmh}^{E}}
\]
\[ IF = \frac{R_{\text{cat}} - 2R_{\text{mph}}}{2R_{\text{mph}}} \]  

(7)

It should be noted that \( IF \) can also be calculated considering the wave resistance, but then the total resistance must be decomposed using some decomposition method. In order to obtain the wave resistance, form factor should be known for both catamaran configurations and the monohull. In [5] the form factor is assumed to be identical for both catamaran configurations and the monohull. To avoid error caused by this assumption, \( IF \) in this paper is calculated on the basis of the total resistance.

\( IF \) is calculated for experimentally and numerically obtained results using fine grid G2. Summarized results are given in Table 3 for both catamaran configurations. The curve of \( IF \) as a function of \( Fn \) is shown in Figure 5. Even though there are discrepancies between experimentally and numerically obtained \( IF \), both curves show the same trend. As it can be seen from Table 3 and Figure 5, values of \( IF \) are higher for the narrower catamaran configuration except for \( Fn=0.4 \). This is in accordance with results obtained in [7] and [11]. The minimum of \( IF \) curve shifts towards higher values of \( Fn \) for narrower catamaran configuration C1. Value of \( Fn \) where this minimum occurs is important because at this particular value interference resistance is smallest, even negative for C2.

**Table 3:** Comparison of experimentally and numerically obtained \( IF \)

<table>
<thead>
<tr>
<th>( Fn )</th>
<th>( IF_{\text{EXP}} )</th>
<th>( IF_{G2} )</th>
<th>( IF_{\text{EXP}} )</th>
<th>( IF_{G2} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.3</td>
<td>0.2560</td>
<td>0.1762</td>
<td>0.0499</td>
<td>0.0001</td>
</tr>
<tr>
<td>0.35</td>
<td>0.0933</td>
<td>0.0525</td>
<td>-0.0040</td>
<td>-0.0359</td>
</tr>
<tr>
<td>0.4</td>
<td>0.0306</td>
<td>0.0186</td>
<td>0.2675</td>
<td>0.2582</td>
</tr>
<tr>
<td>0.5</td>
<td>0.3583</td>
<td>0.3499</td>
<td>0.1221</td>
<td>0.1238</td>
</tr>
</tbody>
</table>

**Figure 5:** The curve of \( IF \) as a function of \( Fn \)
Figure 6 illustrate the wave profile along the monohull and C1 for $F_n=0.5$. This figure shows only starboard side of these two models. The significant change in the wave profile along the catamaran demihull as a result of the interference between the demihulls can be noticed.

![Figure 6: Wave profile along the monohull (upper) and C1 (lower) for $F_n=0.5$](image)

Wave patterns of S60 monohull obtained with fine grid G2 for $F_n=0.3$ and $F_n=0.5$ are shown in Figure 7. It can be noticed that wave elevations are more than two times higher for $F_n=0.5$.

![Figure 7: Wave patterns of S60 monohull for $F_n=0.3$ (left) and $F_n=0.5$ (right)](image)

The obtained hydrodynamic pressure distribution on the starboard side and portside of C1 configuration for $F_n=0.4$ can be seen in Figure 8. As it was expected, pressure differences are significantly larger on the starboard side. Wave patterns for both catamaran configurations obtained using fine grid G2 for two $F_n$ values are shown in Figure 9. It can be noticed that wave elevations for narrower catamaran configuration are significantly higher than the ones for wider catamaran configuration. The first wave crest behind the monohull and two catamaran configurations is closer to the stern for smaller values of $F_n$. Also, as it can be seen
from Figure 9, the first wave crest is closer to the stern in the case of narrower catamaran configuration.

**Figure 8**: Hydrodynamic pressure distribution on the starboard side (upper) and the portside (lower)

**Figure 9**: Wave patterns of C1 (left) and C2 (right) for two $Fn$ values
5 CONCLUSION

In this paper, numerical simulations of the viscous flow around the S60 monohull and two catamaran configurations are performed in order to investigate the interference resistance and its dependence on the spacing between hulls. Interference resistance is examined through $IF$. The total resistance obtained with numerical simulations show satisfactory agreement with experimental results, although all values are overestimated. The greatest relative deviations are obtained for lower $Fn$ values for the monohull. The uncertainty of measuring resistance for such a small model can be relatively large at lower values of $Fn$. Despite overestimations, the same trend of experimentally and numerically obtained $IF$ curves can be noticed. Therefore, it can be concluded that CFD is a very useful tool in catamaran preliminary design. The results point out that interference effects are more significant for narrower catamaran configuration and that the minimum of $IF$ curve shifts towards higher values of $Fn$. Also, obtained wave elevations are significantly higher for narrower configuration. The computation data represent a good basis for future studies involving the influence of trim and the arrangement of hulls in catamaran configuration on the interference resistance. Inclusion of some additional goals in finding optimal spacing between hulls, for example seakeeping characteristics, will be investigated in future work.

REFERENCES

[11] Zaghi, S., Broglia, R., Di Mascio, A. Analysis of the interference effects for high-


NUMERICAL SIMULATION OF LARGE COMMERCIAL SHIP NAVIGATION ON PARANÁ RIVER, ARGENTINA

MARINE 2017

GERARDO FRANCK, SILVINA MANGINI, HÉCTOR PRENDE, JOSÉ HUESPE AND YASSER PALAY ESQUIVEL

Facultad de Ingeniería y Ciencias Hídricas
Universidad Nacional del Litoral
Ciudad Universitaria, 3000 Santa Fe, Argentina
e-mail: gerardofranck@yahoo.com.ar, silvinamangini@yahoo.com.ar
web page:http://fich.unl.edu.ar

Key words: Computational Methods, Post-Panamax sailing, Argentinean waterway

Abstract. In recent years, the large commercial ships (Post-Panamax) started sailing the Argentinean waterway of the Paraná River. So that, the research project of the FICH-UNL called "Analysis of Hydro-Sedimentological Effects Caused by River Navigation on Argentinean Waterways" decided to use the computational simulation as one of tools research to study the hydraulic and sedimentological effects caused by the navigation of these ships on the morphology of the rivers of the Argentine waterway.

One of the premises of this work was to explore the capabilities of the Adapco STAR CCM + CFD code to simulate the ship navigation in calm water in order to measure the heights of waves generated (among others), validating its results with the experimental one (physical model of large ships navigation carried out at the Hydraulics Laboratory of FICH-UNL).

Navigation situation simulated: JAPAN Bulk Carrier (JBC; Lpp: 280 m, Bwl: 45 m and D: 25 m) with a service speed of 22 knots (11.3m/s) navigating against a flow stream with velocity of 2m/s (habitual situation of JBC navigation on Paraná River).

The numerical modeling reproduced the navigation ship in scale (1/36.7), using a Computational Fluid Dynamics code (CFD) by finite-volume method (VOF). A study of mesh dependence was performed to analyze the convergence model. A mesh of hexahedral cells was used in the Volume of Fluid (VOF) Multiphase Model for waves and dumping waves. The turbulence was modeled by a k-Epsilon model from Reynolds Average. Wave damping was used in the lateral walls of the domain.

1 INTRODUCTION

This work presents one of the first tasks carried out within the Research Project of the FICH-UNL (Facultad de Ingeniería y Ciencias Hídricas, Universidad Nacional del Litoral) called "Analysis of Hydrosedimentological Effects Caused by River Navigation in Argentinean Waterways". The main purpose of this project is to quantify the hydraulic and sedimentological effects caused by ships navigation on the morphology of the rivers of Argentine waterway.

The waves and induced currents produced by the navigation of ships produce an increase of turbulence levels, fluctuating overpressures, shear forces and bed sediment transport, which
affect the natural morphology of the rivers and navigation channels.

The research focused on the development of four tasks, with the purpose of knowing the wave pattern and the maximum height of the wave produced by the ships navigation on the Argentine Waterway:

1- Study of existing expressions or, generation of new ones.
2- Development of a sensor to measure wave heights.
3- Physical and experimental simulation of Barge Train and Pos-Panamax ship at the Hydraulic Laboratory of FICH.
4- Numerical simulation of Pos-Panamax ship navigation in order to estimate the heights of waves generated (among others), since large ships are just beginning to navigate the Argentine waterway.

In recent years, computational techniques of fluid dynamics as CFD (Computational Fluid Dynamics), have been incorporated into optimization of simulation procedures for ship’s hull configuration, making the CFD simulations, an important tool for ship design and performance analysis.

Currently, we do not have empirical expressions to estimate the maximum wave height (Hm) for fast ships, such as the JBC, (situations with velocity and Froude number greater than 20 knots and 0.7 respectively), because the existing ones (Pianc and Sorensen, that are adaptable to navigational situations in our rivers), are valid for Froude numbers up to 0.7. Therefore, we try to incorporate the CFD as a tool to predict these behaviors and, at the same time, be able to explore the capabilities of the Adapco STAR CCM + CFD code on the numerical simulation of the JBC navigation in calm water.

The main purpose of this numerical simulation was to reproduce the wave pattern, from which, it can be estimated height of waves generated by the fast navigation of the JBC, at different distances from the navigation line.

In the present work, are presented one of the first results of the numerical simulation of unsteady turbulent flow around a fast ship (JBC), navigating in calm (state that is an exception in the case of marine situations) and deep water. Even though, the deep water conditions used in this simulation do not faithfull represents the characteristics of navigation in rivers, it facilitates the simulation, achieving stable, with good results in terms of the goal pursued. Numerical analyses are done by means of RANS solution and the well-known standard $k-\varepsilon$ two equation turbulence model was used as turbulence model.

The validation of the numerical modeling was made from physical and experimental model results of the same navigation situation

2 NUMERICAL MODELLING

2.1 Ship geometry and conditions

In this study, it was simulated the navigation of JBC Bulk Carrier (Figure 1) on calm water in a scale of 1/36.7. The numerical simulated situation was a JBC with a service speed of 22 knots (11.3 m/s) navigating against a flow stream with velocity of 2 m/s (Habitual situation of JBC navigation on Paraná River). The main properties of the geometry are depicted in Table 1. Service speed of JBC in numerical simulation and full scale and, their correspond ship’s Froude number for the JBC navigation are shown in Table 2.
Table 1: Main properties of JBC geometry

<table>
<thead>
<tr>
<th>Main particulars</th>
<th>Full scale</th>
<th>Model scale (1:36,7)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length between perpendiculars</td>
<td>L_PP (m)</td>
<td>280</td>
</tr>
<tr>
<td>Length of waterline</td>
<td>L_WL (m)</td>
<td>285</td>
</tr>
<tr>
<td>Maximum beam of water line</td>
<td>B_WL (m)</td>
<td>45</td>
</tr>
<tr>
<td>Depth</td>
<td>D (m)</td>
<td>25</td>
</tr>
<tr>
<td>Draft</td>
<td>T (m)</td>
<td>14</td>
</tr>
<tr>
<td>Displacement volume</td>
<td>V (m³)</td>
<td>151369</td>
</tr>
<tr>
<td>Block coefficient (CB)</td>
<td>∇ / (L_PP B_WL T)</td>
<td>0.8581</td>
</tr>
</tbody>
</table>

Table 2: Service speed and Froude ship number (Fn_ship)

<table>
<thead>
<tr>
<th></th>
<th>U: Ship velocity (m/s)</th>
<th>V: Velocity current (m/s)</th>
<th>Fn_ship</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full scale</td>
<td>11.3</td>
<td>-2</td>
<td>0.253</td>
</tr>
<tr>
<td>Numerical Model Scale</td>
<td>1,86</td>
<td>-0.33</td>
<td>0.253</td>
</tr>
</tbody>
</table>

2.2 Governing equations

For incompressible flows without body forces, the averaged continuity and momentum equations may be written in tensor form and Cartesian coordinates as follows [3]:

\[
\frac{\partial \rho \vec{u}_i}{\partial x_i} = 0; \quad \text{for the continuity} \tag{1}
\]

\[
\frac{\partial (\rho \vec{u}_i)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho \vec{u}_i \vec{u}_j + \rho u_i u_j \right) = \frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}; \quad \text{for the momentum equations} \tag{2}
\]

in which \( \tau_{ij} \) are the mean viscous stress tensor components, as shown in Equation (3)

\[
\tau_{ij} = \mu \left( \frac{\partial \vec{u}_i}{\partial x_j} + \frac{\partial \vec{u}_j}{\partial x_i} \right) \tag{3}
\]

Where \( P \) is the mean pressure, \( \vec{u}_i \) is the averaged components of the velocity vector, \( \left( \rho u_i, u_j \right) \) is the Reynolds stresses, \( \mu \) is the dynamic viscosity and \( \rho \) is the fluid density. To model the fluid flow, it was used a solver employed with finite volume method, which discretizes the integral formulation of the Navier–Stokes equations. The RANS solver employs a predictor–corrector approach to link the continuity and momentum equations. The eddy viscosity \( \nu_t \) is calculated by combining turbulent kinetic energy, \( k \), and the rate of dissipation of the turbulent energy \( \varepsilon \). The standard \( k-\varepsilon \) two equation turbulence model has been used to simulate the turbulent flows. Where turbulent Prandl numbers used \( \sigma_k = 1 \) \( \sigma_{\varepsilon} = 1.2 \); and \( C_{\mu} = 0.09; C_{\varepsilon} = 1.44 \) and \( C_{\mu 2} = 1.9. \)
2.3 Physics modelling

A standard k–ε model was used as the turbulence model (extensively used for many industrial applications [1] and considered as economical in terms of CPU time [5]). The “Volume of Fluid” (VOF) method was used to model and position the free surface, either with a flat or regular wave. The VOF method is considered as “a simple multiphase model, very convenient to simulating flows of several immiscible fluids on numerical grids capable of resolving the interface between the mixtures phases” [1]. This model has high numerical efficiency and suitable for simulating flows in which each phase forms a large structure, with a low contact area between them. The VOF model is based on the assumption that the same basic governing equations used for a single phase problem can be solved for all the fluid phases of the domain, assuming the same velocity, pressure and temperature. Therefore, the equations are solved for an equivalent fluid whose properties represent the different phases and independent volume fractions [1]. The input velocity and volume fraction of both phases in each cell and the output pressure are all functions of the plane wave or regular wave used to simulate the free surface. The free surface is not fixed; it is dependent on the specifications of this flat or regular wave, with the VOF model making calculations for both the water and air phases. In this work, a second-order convection scheme was used throughout all simulations in order accurately capture sharp interfaces between the phases. In the Figure 2, the free surface was represented by showing the water volume fraction profile on the hull, a value of 0.5 for the volume fraction of water implies that a computational cell is filled with 50% water and 50% air. This value therefore indicates the position of the water–air interface, which corresponds to the free surface.

![Image](image_url)

Figure 2: Free Surface representation on the ship hull

In this work, the segregated flow model, which solves the flow equation in an uncoupled manner, was applied throughout all simulations in the RANS solver. The grid is must be simply refined in order to enable the variations in volume fraction to be more accurately captured.

2.4 Computational domain and boundary condition

The computational domains were created mainly to see the waves producing by the ship navigation in calm water. In the domains, an overset mesh was used to facilitate the motions of the full-scale ship model.
The determination of these boundary conditions is of critical importance to obtain accurate solutions. The selection of the appropriate boundary conditions can prevent unnecessary computational costs when solving the problem [2].

The boundary conditions in the domain for this simulation (calm water) were the ones depicted in Figure 3. The computational domain extends from $-2.2 \text{ Lpp} < x < 1.2 \text{ Lpp}$, $0 < y < 1.5 \text{ Lpp}$ and $-2.25 \text{ Lpp} < z < 1.15 \text{ Lpp}$, where half of the entire computational domain was taken into account due to vertical plane symmetry. The ship axis is located along the $x$-axis, with the bow located at $x = \text{Lpp}$ and the stern at $x = 0$. The still water level lies at $z = 0$.

A general view of the computation domain with the JBC hull model and the notations of selected boundary conditions are depicted in Figure 3.

In order to reduce computational complexity and demand, only half of the hull is represented. A symmetry plane forms the centerline domain face. The velocity inlet (-2.19 m/s) boundary condition was set in the negative direction $x$ and the positive $x$ direction was modeled as pressure outlet.

The top and bottom boundaries were both selected as velocity inlet. The symmetry plane has a symmetry condition, and the side of the domain (the negative $y$ direction) has a velocity inlet boundary condition as well. The use of the velocity inlet boundary condition at the top and the side of the background prevents the fluid from sticking to the walls, so this prevent a velocity gradient from occurring between the fluid and the wall. Hence, the flow at the very top and very side of the background is also directed parallel to the outlet boundary, this enables to be prevented fluid reflections from the top and side of the domain. The selection of the velocity inlet boundary condition for the top and bottom (deep water and infinite air conditions), do not faithfull represents the characteristics of navigation in rivers, however facilitate the simulation, achieving a stable simulation, with good results in terms of the objective pursued in this study (obtaining a wave pattern and height of waves, see Validation and Verification).

The waves generated by the presence of the JBC were treated by applying a numerical damping beach with a damping length of 5m (VOF wave damping capability of STAR-CCM+). This numerical beach model was used in downstream and transverse directions to
prevent reflections effects of waves. The pressure outlet boundary condition was set behind the ships to prevent back flow occurring and fixes static pressure at the outlet.

2.5 Mesh Generation

Mesh generation was performed using the automatic meshing facility in STAR-CCM+, which uses the Cartesian cut-cell method. A trimmed cell mesher was employed to produce a high-quality grid for complex mesh generating problems. The ensuing mesh was formed primarily of unstructured hexahedral cells with trimmed cells adjacent to the surface.

The computation mesh has areas of progressively refined mesh size in the area immediately around the hull, in the expected free surface and in the wake produced by the ship, in order to capture the complex flow features appropriately. Volumetric controls were using to refine the mesh density in these zones. The mesh was rigid, unstructured and body-fixed; in order to the motions of the body agree to the movement of grid points. The most refined mesh areas around the hull remained within the boundaries of the overset domain. The mesh had areas with refined progressively mesh size, around the hull, on the expected free surface, near the stern, near the bow and in the wake produced by the ship navigation. These refined mesh area allow that the complex flow features were appropriately captured. When generating the volume mesh, an extra care was given to the overlapping zone between the background and overset regions.

It is necessary to do a grid refinement study in order to estimate the grid related uncertainty. Therefore, for the numerical solution of the governing equations the domain was discretized in two different resolutions as coarse (807137) and fine (3,091,741 still running). The numerical grid system employed for the coarse grid with a general view of the computational domain and overset regions is shown in Figure 4. The coarse mesh grid had a range of cell sizes between 0.007 mts (refined grid) to 0.45 mts.

![Figure 4](image)

Figure 4: A general view of the background and overset regions and the applied boundary conditions.
Figure 5a shows a cross-section of the computation mesh where it can see clearly the refinement to capture the Kelvin wake. Figure 5b shows the surface mesh on the hull.

![Cross-section of the computation mesh showing with refined mesh to capture the Kelvin wake.](image)

![Surface mesh on the hull.](image)

**Figure 5:** a: Cross-section of the computation mesh showing with refined mesh to capture the Kelvin wake. b: Surface mesh on the hull.

### 2.6 Choice of the time step

The Courant number (CFL), which is the ratio of the physical time step (Δt) to the mesh convection time scale, relates the mesh cell dimension Δx to the mesh flow speed U as: 

\[ \text{CFL} = \frac{U \Delta t}{\Delta x} \]

The Courant number is typically calculated for each cell and should be less than or equal to 1 for numerical stability.

Often, in implicit unsteady simulations, the time step is determined by the flow properties, rather than the Courant number. For resistance computations in calm water, the time step size is determined by \( \Delta t = 0.005\text{–}0.01L/U \) (where \( L \) is the length between perpendiculars) in accordance with the related procedures and guidelines of [5]. The Navier-Stokes equations were discretized by mean of first-order temporal scheme. The physical time step choiced was 0.04 seconds.

### 2.7 Wall Functions treatment

The wall functions are a set of semi-empirical functions used to satisfy the physics of the flow in the near wall region. Turbulence is affected in many ways by the presence of the wall through the non-slip condition that must be satisfied at the wall. Four areas in the near wall region are defined, the laminar sub-layer, the blending region, the log law region and the outer region. Each region has a different effect on turbulence and a particular care must be taken to the \( y^+ \) position of the first cell in the boundary layer. A different set of equations will be used depending on the size of this cell but however this one must not be comprised between \( y^+ = 5 \) and \( y^+ = 30 \) because no turbulent model is available in this area. Instead of not resolving the entire boundary layer for a \( y^+ \) comprised in the viscous sub-layer and buffer layer, wall functions are used to bridge the viscosity-affected region between the wall and the fully-turbulent region, [1].

The All-\( y^+ \) wall treatment is a hybrid treatment that attempts to emulate the high- wall treatment for coarse meshes and the low-\( y^+ \) wall treatment for fine meshes. It is also...
formulated with the desirable characteristic of producing reasonable answers for meshes of intermediate resolution (that is, when the wall-cell centroid falls within the buffer region of the boundary layer), [1]. Two-Layer All $y^+$ Wall Treatment is a formulation that is identical to the All $y^+$ Wall Treatment, but contains a wall boundary condition for $\varepsilon$ that is consistent with the two-layer formulation. This treatment is the recommended wall treatment where provided [1].

All the $y^+$ values for the coarse grids can be seen on Figure 6 and the number of grid points and average $y^+$ values are given in Table 3.

<table>
<thead>
<tr>
<th>Grid points number</th>
<th>average $y^+$</th>
</tr>
</thead>
<tbody>
<tr>
<td>807137</td>
<td>104.03</td>
</tr>
</tbody>
</table>

3 RESULTS

The following section will outline the simulation results achieved during this study, and will also provide some comparison with experimental results.

3.1 Wave Pattern

Figure 7 shows the volume fraction of air plot and the wave elevation at the bow and at the stern produced by the navigation of JBC, after the simulation has completed its run (solution time: 152 seconds, CPU time: 98.3 hours).
Figure 8 shows both a plant view of global wave pattern generated by the JBC navigation, (after the simulation has completed its run) and the locations of longitudinal cuts of the wave pattern (isosurfaces scene) from which were obtained the longitudinal profiles of waves.

![Figure 8: Wave pattern after the simulation has completed its run. Scheme of wave pattern cut to obtain longitudinal profiles of waves at 100mts of JBC hull edge.](image)

Figure 9 shows the longitudinal profiles of water free-surface deformation at JBC hull contour and the longitudinal profile of wave cut at 100 m from the hull edge. In this Figure, can be appreciated in each profile, the maximum deformation of water free-surface at the contour hull and the maximum amplitude of the wave (Hm) at 100 m, which were estimated from the position (z in m) indicated by the color scale.

![Figure 9: Logitudinal profile of water free-surface deformation at JBC hull contour and longitudinal profiles of waves at 100 mts from the JBC hull edge.](image)
Table 4 shows the maximum amplitude of water free-surface deformation at JBC hull contour and the maximum waves height (Hm) at 100 m from JBC hull edge produced by the JBC navigation in the simulation model and its corresponding value in full scale.

Table 4: Maximum water free-surface deformation at ship contour and maximum waves height (Hm) at 100 m from JBC hull edge.

<table>
<thead>
<tr>
<th></th>
<th>Maximum water free-surface deformation at JBC hull contour</th>
<th>Hm at 100 m</th>
</tr>
</thead>
<tbody>
<tr>
<td>Numerical Model (E: 1:36.7)</td>
<td>0.39716 m</td>
<td>0.0508 m</td>
</tr>
<tr>
<td>Full scale</td>
<td>14.31 m</td>
<td>1.86 m</td>
</tr>
</tbody>
</table>

4 VALIDATION AND VERIFICATION

The results of the numerical simulation were compared with the results of an experimental physical model on scale (1:127). Based on the similarity of the Flow Froude number between the physical model and full scale (F=0.841), the physical model reproduced the same ship (JBC) navigation condition simulated in the computational model already presented, but, in rivers navigation conditions of Argentine Waterway (Shallow Water). The test was conducted in a laboratory channel of FICH, Figure 10.

On the physical model, waves produced by the ship navigation were estimated (from video images analysis) on three scales placed at distances of 0.79 m (100 m in full scale), 1.58 m (200 m in full scale), and 2.37 m (300 m in full scale), see Figure 10.

Table 5 shows the characteristics of the simulated situation in the physical model and the maximum height of the wave measured at 0.79 m (100 m in full scale) produced by the ship navigation, and, Figure 10 shows images obtained during the experiment.

Figure 10: Images of the physical model during the test performance

Table 5: Characteristics of simulated navigation situation in the physical model.

<table>
<thead>
<tr>
<th>Length (m)</th>
<th>Beam (m)</th>
<th>Height (m)</th>
<th>Draft (m)</th>
<th>Depth (m)</th>
<th>Ship velocity (m/s)</th>
<th>Scale location from the ship edge (m)</th>
<th>Height measured on the scale. Hmax (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model</td>
<td>Full Scale</td>
<td>Model</td>
<td>Full Scale</td>
<td>Model</td>
<td>Full Scale</td>
<td>Model Full Scale</td>
<td>Model Full Scale</td>
</tr>
<tr>
<td>2.2</td>
<td>280</td>
<td>0.356</td>
<td>45</td>
<td>0.196</td>
<td>25</td>
<td>0.11</td>
<td>14</td>
</tr>
</tbody>
</table>
In Figure 11, a perspective view shows good match between the bow wave patterns around the ship hull of computational and physical models. Both models images also show a similar thin water elevation close to the bow.

![Figure 11. Perspective view of the bow wave patterns of computational and experimental models. (Fn_{ship} = 0.253)](image)

For the scale of numerical simulation, the theoretical dynamic height of the water at the ship bow is given by: $V^2/2g$ (since the ship remains still facing a flow with a given inlet velocity). Being the flow velocity inlet of -2.19 m/s, the theoretical dynamic height results of 24.4 cm. The maximum height of water at the ship bow in the numerical simulation resulted of 23.705 cm (see Figure 12).

![Figure 12. Height of water at the bow at the end of computational simulation.](image)
5 CONCLUSIONS

The comparison of results between computational and experimental simulation of JBC navigation, for the simulated navigation conditions ($F_{n,\text{ship}} = 0.253$), shows that:

- Star CCM + predicts satisfactorily the free surface flow around the ship hull, navigating in shallow waters (Paraná Waterways), even though, deep water boundary conditions are used in the simulation.

- The maximum wave height at 100 m from the hull side, obtained from computational model ($H_m = 1.86$ m in full scale) and experimental model ($1.94$ mts in full scale) were very close (error: 4%).

- An error of 2.8% arises from the comparison between the theoretical dynamic height of water at the ship bow and the height of water observed at the ship bow in the numerical simulation.

For the above reasons, the numerical simulation (STAR CCM+ CFD) could be considered as a useful tool to predict free surface flows (waves) generated by ship navigation. Even using a coarse mesh (807139 cells), the comparison of results between both models (numerical and physical) was acceptable.

It is suggested to perform new numerical and experimental simulations of JBC navigation at different velocities, and, use finer for numerical simulation.

The computational simulation with a finer mesh (3,091,741 cells) for the same navigation situation of JBC is actually running.

6 REFERENCES


[2] Date, J.C., Turnock, S.R., 1999. A Study into the Techniques Needed to Accurately Predict Skin Friction Using RANS Solvers with Validation Against Froude's Historical Flat Plate Experimental Data. University of Southampton, Southampton, UK p. 62 (Ship Science Reports, (114))


NUMERICAL STUDIES ON AIR RESISTANCE REDUCTION METHODS FOR A LARGE CONTAINER SHIP WITH FULLY LOADED DECK-CONTAINERS IN OBLIQUE WINDS

T. V. NGUYEN 1,3, *, N. SHIMIZU 2, A. KINUGAWA 2, Y. TAI 2, AND Y. IKEDA 1

1 Research Organization for the 21st Century (RO-21)
Osaka Prefecture University
1-1 Gakuen-cho, Nakaku, Sakai, Osaka 599-8531, Japan
e-mail: dx102001@edu.osakafu-u.ac.jp, web page: http://www.osakafu-u.ac.jp/

2 Imabari Shipbuilding Co. LTD.
30 Showa-cho, Marugame, Kagawa 763-8511, Japan
e-mail: shimizu.nobuyuki@imazo.com, web page: http://www.imazo.co.jp/

3 Faculty of Transportation Mechanical Engineering
University of Science and Technology-The University of Danang
No. 54 Nguyen Luong Bang St., Lien Chieu Dist., Danang, Vietnam
email: nvtrieu@dut.udn.vn, web page: http://www.dut.udn.vn/

Key words: Numerical Method, CFD, Air Resistance, Container Ship

Abstract. In this study, the aerodynamic characteristics of the complex air flows acting on a large container ship model are numerically investigated by using a commercial CFD code. The main target is to reduce the air resistance in oblique winds, especially at the small angle of wind directions. Some container side-covers with different size and location are developed to shut the gap flow, and a center wall is also applied. The numerical results show that the air resistance at 30 degrees of wind direction can be reduced significantly up to 50%, 30% and 15% by full side-covers, front-half side-covers and lower front-half side-cover, respectively.

1 INTRODUCTION

Now a day, the newly designed container ships become very large and the number of containers on deck has increased rapidly. Consequently, the air resistance acting on the ship also becomes larger and the maximum value usually occurs at 20-40 degrees of wind direction conditions. Therefore, reducing the air resistance of such ships in oblique winds must be paid attention in order to reduce the fuel consumption and CO₂ emission from the international shipping.

Ouchi [1] studied on air resistance reduction of a large container ship in the sea. Their study showed that covering all longitudinal gaps among container blocks has a great effect on decreasing the longitudinal resistance around 60 degrees of wind direction.

Kim [2] studied on superstructure modification for air resistance reduction of a large container ship. Gap-protector between containers stacks and a visor in front of the upper deck are found to be the most effective ways for air resistances reduction in wide range of heading
angle of winds.

Recently, Watanabe [3] & Nguyen [4] numerically and experimentally studied on the air resistances reduction methods for a 20,000TEU container ship model with full containers on deck. They developed different types of apparatuses, which can reduce the air resistance.

In this study, the aerodynamic characteristics of the complex air flows acting on the same large container ship model as that using in their studies are numerically investigated by using a commercial CFD code. The main target is reducing the air resistance in oblique winds, especially at the small angle of wind directions. Effects of a full and some partial side-covers with different size and location as well as a center-wall to shut the gap flow are numerically examined.

2 PRINCIPAL PARTICULARS OF SHIP AND MODEL

The main particulars of the 20000TEU container ship and its scale model are presented in Table 1.

<table>
<thead>
<tr>
<th>Unit</th>
<th>Ship</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scale</td>
<td></td>
<td>1/255.3</td>
</tr>
<tr>
<td>L_{OA}</td>
<td>m</td>
<td>400</td>
</tr>
<tr>
<td>L_{PP}</td>
<td>m</td>
<td>383</td>
</tr>
<tr>
<td>Breadth (B)</td>
<td>m</td>
<td>58.5</td>
</tr>
<tr>
<td>H</td>
<td>m</td>
<td>49.02</td>
</tr>
<tr>
<td>draft</td>
<td>m</td>
<td>14.5</td>
</tr>
<tr>
<td>A_F</td>
<td>m^2</td>
<td>2890</td>
</tr>
<tr>
<td>A_S</td>
<td>m^2</td>
<td>18000</td>
</tr>
</tbody>
</table>

The side profile and frontal shape of the ship model can be seen in Fig. 1.

3 AIR RESISTANCE REDUCTION METHODS

In previous studies [1]–[4] it has been already confirmed that the gaps between container
blocks on deck increase the air resistance acting on container ships. It was reported that covering the all gaps by using the full side covers [1] or gap protectors [2] can significantly reduce the air resistance. However, in practical, the very large and heavy side-covers increase the ship’s weight. In this study, the effects of two kinds of partial side-covers with lighter weight on the reduction of air resistance are numerically examined. In the previous studies of the authors, the gap flows of the front part of the ship considerably generate the wake and increase the air resistance in head winds [4]. Therefore, partial covers at the front part of deck containers can be effective even in oblique winds.

Another alternative is a center wall, which may be, shut the horizontal gap flow at the middle of each gap.

The ship models with the full side-covers, the front side-covers, the front-half side-covers and the center wall can be seen in Figs. 2~5.

![Figure 2: Full side-cover](image1)

![Figure 3: Front side-cover](image2)

![Figure 4: Center wall](image3)

![Figure 5: Front-half side-cover](image4)

4 COMPUTATIONAL FLUID DYNAMICS

A CFD commercial code, ANSYS Fluent V14.5 is used to solve the RANS equations. The computational domain, coordinate system, mesh generation, and numerical setup are discussed in the following sections. The calculated results were evaluated by comparing the results with experimental results of the ship without and with full side-covers [3].

4.1 Computational Domain

The computational domain is determined as shown in Fig. 6 according to check the
accuracy with the experimental results [3].

Figure 6: Computational domain

4.2 Coordinate System and Coefficients

This study uses the same coordinate system as the experimental measurement, which has been carried out by Watanabe [3]. The coordinate is shown in Fig. 7. The aerodynamic coefficients, $C_X$, $C_Y$, and $C_N$ are defined as Equations 1~3.

The aerodynamic coefficients, $C_X$, $C_Y$, and $C_N$ are defined as Equations 1~3.

$$C_X = \frac{X_A}{\frac{1}{2} \rho_A U^2 A_F}$$

$$C_Y = \frac{Y_A}{\frac{1}{2} \rho_A U^2 A_S}$$

$$C_N = \frac{N_A}{\frac{1}{2} \rho_A U^2 A_S L_{OA}}$$

Where: $C_X$, $C_Y$, $C_N$ are a longitudinal force (air resistance), side force, yaw moment coefficients, respectively, $\rho_A$ is air density (kg/m$^3$), and $U$ is velocity at the free stream (m/s).

Figure 7: Definition of coordinates [3]
4.3 Mesh Generation

The computational domain is discretized into the mesh cells. Meshing is an important step because of the mesh quality significantly affects on the numerical results. In this study, due to the complexity of the model, the unstructured mesh is generated in ANSYS Meshing module rather than try to obtain the structured mesh. The total number of the mesh is approximate 4.6 million tetrahedral cells. The maximum skewness is less than 0.9. The dimensionless value of the first mesh, $y^+$, is less than 180 for the k-epsilon model. The cut mesh near the model surfaces is shown in Fig. 8.

![Figure 8: Cut mesh near the surfaces at y=0 (upper), and z=0.12m (lower)](image.png)

4.4 Solution Setup

The k-epsilon turbulence model is used. The standard wall function is applied for near wall treatment. The velocity inlet and pressure outlet are applied for inlet and outlet boundaries, respectively. A no-slip wall condition is applied to the ship model surfaces, and slip conditions are applied to the top, bottom, and sides of the domain. The second order upwind is selected for the momentum, turbulent kinetic energy, and turbulent dissipation rate to increase the solve accuracy.

The aerodynamic coefficients can be regarded to be independent of the wind speed and free stream turbulence intensity [2]. Therefore, the inflow speed is fixed at 10 m/s, and 5% is selected for the turbulent intensity.

All the setup parameters are listed in Table 2.
Table 2: Solution setup for CFD

<table>
<thead>
<tr>
<th>Solver</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Pressure-Based</td>
</tr>
<tr>
<td>Velocity formulation</td>
<td>Absolute</td>
</tr>
<tr>
<td>Time</td>
<td>Steady</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Models</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Viscous model</td>
<td>k-epsilon (2 eqs)</td>
</tr>
<tr>
<td>k-epsilon Model</td>
<td>Standard</td>
</tr>
<tr>
<td>Near-Wall Treatment</td>
<td>Standard Wall Function</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Materials</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid</td>
<td>Air</td>
</tr>
<tr>
<td>Density</td>
<td>1.225 (kg/m³)</td>
</tr>
<tr>
<td>Viscosity</td>
<td>1.7894e-05</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Boundary Conditions</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Velocity inlet:</td>
</tr>
<tr>
<td></td>
<td>Velocity Magnitude: 10 (m/s)</td>
</tr>
<tr>
<td></td>
<td>Turbulent Intensity: 5%</td>
</tr>
<tr>
<td></td>
<td>Turbulent Viscosity Ratio: 10</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure outlet:</td>
</tr>
<tr>
<td></td>
<td>Backflow Turbulent Intensity: 5%</td>
</tr>
<tr>
<td></td>
<td>Backflow Turbulent Viscosity Ratio: 10</td>
</tr>
<tr>
<td>Ship</td>
<td>No-slip wall</td>
</tr>
<tr>
<td>Top, bottom, sides</td>
<td>Slip wall</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Solution Methods</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure-Velocity Coupling</td>
<td>SIMPLE</td>
</tr>
<tr>
<td>Scheme</td>
<td>SIMPLE</td>
</tr>
<tr>
<td>Spatial Discretization</td>
<td></td>
</tr>
<tr>
<td>Gradient</td>
<td>Least Square Cell Based</td>
</tr>
<tr>
<td>Pressure</td>
<td>Standard</td>
</tr>
<tr>
<td>Momentum</td>
<td>Second Order Upwind</td>
</tr>
<tr>
<td>Turbulent Kinetic Energy</td>
<td>Second Order Upwind</td>
</tr>
<tr>
<td>Turbulent Dissipation Rate</td>
<td>Second Order Upwind</td>
</tr>
</tbody>
</table>

5 RESULTS AND DISCUSSIONS

In the previous papers [3] [4], the aerodynamic forces calculated by the CFD code in the same manner as in the present paper were compared with the experimental ones and showed a fairly good agreement with the experimental ones in all wind directions. Therefore, the accuracy of the CFD simulation was evaluated.

On the basis of the CFD results, the effects of the full side-covers on the velocity and pressure distribution, velocity vector and streamline again are discussed, and the effects will be compared with those of the front side-covers, the front-half side-covers, and the center-wall.
5.1 Velocity and Pressure

The velocity and pressure distribution on the horizontal plane at z=0.12m are shown in Figs. 9-11. The z denotes the distance between the plane and the water surface, and the plane is located at near middle height of the deck containers as shown in Fig. 1.

The velocity and pressure distributions around the deck containers of the ship model without covers are shown in Fig. 9. The velocity distribution shows that very low velocity regions are created at the bow starboard region as well as behind the rear container blocks. The pressure distribution shows that high pressure acts on each corner of the containers. The pressure causes the air resistance increase by gap flows.

The velocity distributions around the deck containers with the three kinds of side-covers and the center-wall wake shown in Fig. 11 demonstrate that the flows around the containers are changed by the covers and the wall. The full side-cover and the front side-cover make a high flow at the front corners in starboard side, and the low velocity regions behind the front container blocks disappear. This may be caused by shutting the gap flows. The velocity distributions of the center-wall and the front-half side-covers are similar to that of the model in standard full-loaded condition.

The pressure distributions are shown in Fig. 11 show that high pressure acting on each corner of the deck containers disappears for the full side-covers and the front side-covers. For the case of the center wall, the high pressures at each corner of the gap entrance act as the same as the case of the conventional deck containers without any covers.

![Figure 9: Velocity and distribution of standard model at z=0.12m](image-url)
5.2 Velocity Vectors and Streamlines

The velocity vectors and streamlines took at/from the horizontal plane $z=0.12m$ are presented in Figs. 12 and 13.
Fig. 12 shows that the side-covers shut the flows that across the gaps between deck containers, while the center walls only stop the flows across the center. Nevertheless, the flows can enter or escape the gap from each side. That is the reason why the wake appears on the starboard side of the center walls model.

Fig. 13 show that the full side-covers and the front side-covers reduce the vortex generated at the starboard side, while the front-half side-covers and center-wall models have large wakes at the bow starboard side.

5.3 Aerodynamic Coefficients

In the previous study [4] it has been confirmed that the air resistance in headwinds can be reduced by up to 20% by using apparatuses. The target of the present study is to reduce the air resistance in oblique winds.
The aerodynamic coefficients of the container ship models at wind angle from 0 to 180 degrees are presented in Figs. 14 ~ 16. It should be noted that in the figures the calculated ones up to 30 degrees of wind angle are shown for the cases of the front side-covers and the front-half side-covers.

Fig. 14 shows that at 30 degrees, the maximum reduction of the air resistance, more than 50%, is obtained by the full side-covers, and the front side-covers follow it. The front-half side-covers and the center-walls are not so effective. The reason can be understood from the pressure distribution shown in Fig. 11. The full side-covers shut all the gap flows, then the pressures acting on the frontal side and the back side of each container on deck are almost same. This means that no added air resistance is generated by the gaps.

The results demonstrate that at 60 degrees of wind angle, the full side-covers produce the thrust although the resistances are created for other cases.

In the following wind, the full side-covers generate much smaller thrust than the center walls and standard models.

Figs. 15 and 16 show that the side-covers increases the side force at the wind angle from 60 to 150 degrees, and increases the yaw moment from 30 to 60 degrees and 120 to 150 degrees.

![Figure 14: Longitudinal force coefficients](image-url)
The longitudinal force coefficients, or the resistance coefficient, and its reduction at the wind angle of 0 and 30 degrees are summarized in Table 3. The results in the table show that in a headwind, the side-covers can reduce the air resistance by up to 10~18%. While the center-walls only reduce the air resistance by only about 3%. At 30 degrees of wind angle, the reduction of air resistance is proportional to the size of side covers. They are 50%, 30%, and 15% for the full side covers, front side covers and front-half side covers, respectively. The center-walls shows only 10% reduction.
Table 3: Longitudinal force coefficient and its reduction at 0 and 30 degrees of wind angles

<table>
<thead>
<tr>
<th></th>
<th>Cx</th>
<th>ΔCx (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 degree</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Standard</td>
<td>-0.4926</td>
<td>-</td>
</tr>
<tr>
<td>Full side covers</td>
<td>-0.4063</td>
<td>-17.52</td>
</tr>
<tr>
<td>Front side covers</td>
<td>-0.4326</td>
<td>-12.19</td>
</tr>
<tr>
<td>Front-half side covers</td>
<td>-0.4458</td>
<td>-9.50</td>
</tr>
<tr>
<td>Center walls</td>
<td>-0.4809</td>
<td>-2.37</td>
</tr>
<tr>
<td>30 degrees</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Standard</td>
<td>-0.9012</td>
<td>-</td>
</tr>
<tr>
<td>Full side covers</td>
<td>-0.4204</td>
<td>-53.35</td>
</tr>
<tr>
<td>Front side covers</td>
<td>-0.6313</td>
<td>-29.94</td>
</tr>
<tr>
<td>Front-half side covers</td>
<td>-0.7619</td>
<td>-15.45</td>
</tr>
<tr>
<td>Center walls</td>
<td>-0.8100</td>
<td>-10.12</td>
</tr>
</tbody>
</table>

6 CONCLUSIONS

- The aerodynamic forces acting on a 20000TEU container ship model in the fully loaded deck-containers condition in oblique winds are numerically investigated by using a commercial CFD code.
- The numerical results show that the air resistance is caused by high pressure acting on each corner of the deck container blocks.
- Fully or partially closing the gaps between deck-containers by using side-covers and a center-wall can reduce the air resistance.
- The reduction of air resistance is proportional to the size of side covers.
- The reductions of the air resistance at 30 degrees of wind angle are up to 50%-30%-15%-10% by the full side-covers, front side-covers, front-half side-covers, and the center-wall, respectively.

REFERENCES


SHIP SCALE SELF PROPULSION CFD SIMULATION RESULTS COMPARED TO SEA TRIAL MEASUREMENTS

Vuko Vukčević∗, Hrvoje Jasak, Inno Gatin, Tessa Uroić

University of Zagreb, Faculty of Mechanical Engineering and Naval Architecture,
Ivana Lučića 5, Zagreb, Croatia
e-mail: vuko.vukcevic@fsb.hr, hrvoje.jasak@fsb.hr, inno.gatin@fsb.hr, tessa.uroic@fsb.hr

Key words: CFD, Ship Scale Self Propulsion, Sea Trial Validation, OpenFOAM

Abstract. CFD simulation results for self propelled full scale ship are compared to sea trial measurements in this work. Two–phase RANS based CFD numerical model used in this work is based on the Ghost Fluid Method for numerically robust treatment of discontinuities at the free surface and the algebraic Volume–of–Fluid method for interface capturing. The propeller is modelled as a pressure–jump based actuator disc, allowing CPU time efficient simulations while preserving the accuracy of integral results. The numerical model is implemented in foam–extend, a community driven fork of the OpenFOAM software. The comparison with sea trials includes achieved forward speed, thrust and torque for given shaft speed (in RPM) for a general cargo carrier.

1 INTRODUCTION

As witnessed by many recent workshops [1, 2, 3], CFD is getting increasing attention in ship hydrodynamics. Traditionally, CFD is validated against experiments performed in model scale, mostly due to availability of data. The ability to accurately predict the self–propulsion point of the ship is of great importance in the marine industry, especially considering the current regulations regarding lower greenhouse gas emissions. Model scale CFD studies for a self–propelled ship still represent an active area of research (e.g. [4, 5, 6]), although it has been recently noted by Castro et al. [7] that the flow field at model and full scale is significantly different, especially in the stern area. According to Castro et al. [7], who performed both full and model scale CFD self–propulsion simulations with discretised propeller, the thinner boundary layer at full scale causes a more uniform inflow to the propeller with larger effective advance coefficient, thus increasing the propeller performance.

The availability of data from sea trials is scarce compared to experimental measurements, making a direct comparison of CFD with sea trials difficult. However, there is an ongoing effort for such a direct comparison, as discussed by Ponkratov and Zegos [8, 9], who validated their full scale CFD results against sea trials for a medium range tanker, obtaining good results. Recently, a Workshop on ship scale hydrodynamic computer simulation [10, 11] has been organised by the Lloyd’s Register, providing a unique opportunity for all CFD practitioners to directly compare
their simulation results with sea trial measurements.

The self-propelled ship in CFD can be modelled in a number of ways. The most efficient approach is to model the propeller as an actuator disc, which has been successfully used by many authors (see e.g. Tzabiras et al. [12]). On the other end, fully discretised, rotating propeller with either sliding interface [9] or dynamic overset grids [13, 14] presents the most detailed approach for simulating a self-propelled ship. Since such a detailed approach requires significant computational time, researchers have come up with different ways to speed up their computations without significantly affecting the results. For example, Ponkratov and Zegos [9] first ran their sliding interface CFD simulations with the Multiple Reference Frame (MRF) approach without rotating the propeller until the free surface stabilises, and then turned to full propeller rotation. Also very recently, Carrica et al. [13] presented a partially rotating frame approach, which allowed them to increase the time step with fully discretised propeller by one order of magnitude, while still being able to model a part of the propeller rotation. In this paper, we aim to primarily validate the achieved speed of the ship, where the CPU time efficiency is the most important factor and the detailed flow features behind the propeller are not of primary interest. For this reason, we employ the actuator disc model [15] for the full scale self-propulsion CFD simulations.

The paper is organized as follows. First, the mathematical and numerical model of the two-phase, RANS CFD code navalFoam is presented. The set-up of self-propulsion CFD simulations is presented next, where the results are directly compared to sea trial measurements. Finally, a short conclusion is given outlining the practicality and accuracy of such computations.

2 MATHEMATICAL AND NUMERICAL MODELLING

The CFD model is based on a single-equation formulation for the two-phases: water and air. Water and air are treated as incompressible, where the jump conditions at the free surface are taken into account with the Ghost Fluid Method (GFM). Following Huang et al. [16], GFM is used to discretise the jump conditions at the free surface, yielding interface-corrected discretisation schemes near the interface. For additional details regarding interface-corrected discretisation schemes in arbitrary polyhedral Finite Volume (FV) framework and the details on how they actually model an infinitesimally sharp free surface with respect to pressure and density discontinuity, the reader is referred to Vukčević [17]. The free surface is captured using the algebraic Volume-of-Fluid (VOF) method with interface compression [18], while the turbulence is modelled with a two-equation $k-\omega$ SST turbulence model.

The set of seven coupled, non-linear governing equations is discretised using arbitrary polyhedral FV support in foam-extend-4.0, a community driven fork of the open source software OpenFOAM. All time derivative terms are discretised with first order Euler scheme, while a second order linear upwind scheme is used for the convection term in the momentum equation. The face-centred values are obtained with central differencing with limited non-orthogonal correction for all diffusion terms. The non-linear coupling between equations is resolved in an iterative way using a combination of SIMPLE and PISO algorithms, where two SIMPLE steps encompass six PISO pressure corrections steps within a single time step. No effort was made in this paper to optimise the number of SIMPLE or PISO correctors, which will be the topic for future work.
3 SHIP SCALE SELF PROPULSION SIMULATIONS

The full scale ship considered in this study is the REGAL general cargo carrier [11], where the particulars of the ship are summarised in Table 1. For the purposes of the Lloyd’s Workshop on Ship Scale Hydrodynamic Computer Simulation [11], the ship has been dry-docked, its hull cleaned and propeller polished. The hull, rudder and propeller were 3D laser scanned in order to provide surface meshes to Workshop participants. Further details regarding the quality of hull and appendages can be found in [11]. After cleaning the hull and propeller, the ship was taken to sea trials in ballast condition, where three shaft speeds have been tested and the participants were requested to submit simulation results of achieved speed, thrust, torque, sinkage and trim. In this work, we present a set of results obtained with three grids for a single shaft speed (106.4 RPM) in order to assess numerical uncertainty.

Table 1: REGAL ship’s particulars and trial conditions [11].

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length between perpendiculars</td>
<td>$L_{PP}$, m</td>
</tr>
<tr>
<td>Breadth moulded</td>
<td>$B$, m</td>
</tr>
<tr>
<td>Depth moulded</td>
<td>$D$, m</td>
</tr>
<tr>
<td>Propeller diameter</td>
<td>$D_p$, m</td>
</tr>
<tr>
<td>Service speed at design draught</td>
<td>$V$, kn</td>
</tr>
<tr>
<td>Water density</td>
<td>$\rho_w$, kg/m$^3$</td>
</tr>
<tr>
<td>Kinematic viscosity of water</td>
<td>$\nu_w$, m$^2$/s</td>
</tr>
<tr>
<td>Air density</td>
<td>$\rho_a$, kg/m$^3$</td>
</tr>
<tr>
<td>Kinematic viscosity of air</td>
<td>$\nu_a$, m$^2$/s</td>
</tr>
<tr>
<td>Longitudinal centre of gravity</td>
<td>$LCG$, m</td>
</tr>
<tr>
<td>Vertical centre of gravity</td>
<td>$VCG$, m</td>
</tr>
<tr>
<td>Transverse centre of gravity</td>
<td>$TCG$, m</td>
</tr>
<tr>
<td>Mass</td>
<td>$\Delta$, t</td>
</tr>
<tr>
<td>Pitch radius of gyration</td>
<td>$R_{yy}$, m</td>
</tr>
</tbody>
</table>

3.1 Computational grids

Three grids are generated with cfMesh [19], an open-source mesher which is available in foam-extend, a community driven fork of the open-source software OpenFOAM [20, 21]. Following Workshop instructions [11], the grids extend one $L_{PP}$ in front of the forward perpendicular, two $L_{PP}$ behind the aft perpendicular and one $L_{PP}$ from the starboard, portside and below the baseline. Compared to the real ship geometry, the superstructure and cranes are not modelled in order to keep the grid size to a minimum without significantly affecting the results. The propeller is modelled as a pressure jump-based actuator disc where the radial distribution of the pressure jump is modelled according to [15]. Detail of the fine grid at the stern is presented in Figure 1a, where one can see the circular grid interface of the actuator disc. Figure 1b presents bow stem refinement, while Figure 1c and Figure 1d present Kelvin angle refinement.
and aggressive refinement towards the free surface, respectively.

Three grids consist of approximately 5.6, 7.5 and 11.7 million cells, yielding a refinement ratio of \( r_{mf} = 1.2 \) between medium and fine grid and \( r_{mc} = 1.1 \) between coarse and medium grid. Approximately 95% of the cells are hexahedral, remaining 5% being general polyhedral cells with a small amount (0.03%) of prisms, pyramids and tetrahedra. Maximum non–orthogonality for the fine grid is \( 88.8^\circ \), while the average value is \( 7.3^\circ \). Six layers have been used for the boundary layer, with growth ratio of 1.3. Average dimensionless distance \( y^+ \) at the hull varies from 900 to 1100 between the three grids.

### 3.2 Open water propeller simulations

Full scale simulations of the propeller in open water are carried out to determine thrust and torque curves necessary for the actuator disc model. The two computational grids are created with \texttt{cfMesh}, where the GGI (Generalised Grid Interface) [22] is used to couple non–conformal grids at the rotating interface. The MRF approach is used to model the propeller rotation in a steady–state manner. Thrust and torque curves are obtained by performing five simulations for different advance ratios \( J = \frac{V_a}{(nD_p)} \): 0.2, 0.3, 0.4, 0.5 and 0.6. The advance ratio is varied by changing the advance speed \( V_a \), while keeping the propeller rotation rate \( n = 71.62 \) RPM constant. The open water results are presented in Figure 2a in terms of thrust.
coefficient \( K_T = T/(\rho n^2 D_p^4) \), torque coefficient \( K_Q = Q/(\rho n^2 D_p^5) \) and open water efficiency \( \eta_o = JK_T/(2\pi K_Q) \). Figure 2b presents the vorticity field generated by the propeller for the \( J = 0.4 \) case.

### 3.3 Self–propulsion simulation results

The approximate ship speed for the initial condition is 13 knots according to the Workshop’s instructions [11], modelled as a free stream in this work. Hence, the CFD calculates the difference between prescribed 13 knots and the achieved speed. The ship is free to surge, heave and pitch, while sway, roll and yaw have been constrained. Since a steady state solution is sought, a fixed time step of \( \Delta T = 0.075 \) s has been used, yielding a maximum Courant–Friedrichs–Lewy number of \( CFL = O(10^2) \) during the simulation, where the mean \( CFL \) number is of the order of \( O(10^{-1}) \).

The converged solution on all three grids is achieved after 750 seconds, equivalent to 10000 non–linear iterations. The convergence of forward speed of the ship is presented in Figure 3a, comparing the results with two sea trial runs. Results obtained with three grids fall within the range spanned by sea trials, where the difference between the ISO 15016 value and the CFD result on the fine grid is 0.2%. Moreover, since the oscillatory convergence is achieved with respect to grid refinement, the numerical uncertainty of the forward speed is calculated as:

\[
U_v = 0.5F_s (\max(V) - \min(V)) ,
\]

where \( F_s = 3 \) is the safety factor and \( \max(V) \) and \( \min(V) \) are the maximum and minimum speeds, respectively, achieved with the three grids. Using the results from Figure 3a, the corresponding numerical uncertainty is approximately 0.02 knots, or 0.15% of the fine grid result. Note that the final result oscillates at most by \( \pm 0.0075 \) knots, indicating the degree of iterative uncertainty for the measured ship speed.

The convergence of dynamic trim is presented in Figure 3b, where the calculated values under–predict the measured trim by 0.02 degrees for the first sea trial run, and 0.028 degrees...
Vuko Vukčević, Hrvoje Jasak, Inno Gatin, Tessa Uroič

Figure 3: CFD results compared to sea trials.

for the second. Iterative uncertainty for dynamic trim, calculated with (1) reveals that the uncertainty is 0.025 degrees for the coarse grid, which is lowered to 0.0028 degrees using the fine grid.

The absolute values of resistance and propeller thrust obtained with the fine grid are presented in Figure 4a. Propeller thrust and resistance oscillate with ±8 kN, or ±2.5% compared to the solution averaged over past 200 hundred iterations. In the beginning of the simulation, the resistance is significantly larger than the propeller thrust, which causes the ship to slow down as indicated in Figure 3a. After approximately 2 and a half minutes, the propeller thrust starts to become larger than the resistance, thus causing the acceleration of the ship. Finally, after 10 minutes, propeller thrust and resistance are well balanced and the velocity of the ship does not change significantly (see Figure 3a).

The convergence of resistance with coarse, medium and fine grids is presented in Figure 4b. The convergence is significantly less oscillatory on medium and fine grids compared to the coarse grid. The reason for such strongly oscillatory convergence on the coarse grid is the insufficient grid resolution near the free surface, producing small numerical waves. As can be seen from the zoomed view in Figure 4b, numerical results are insensitive to the mesh refinement, where the iterative uncertainty of approximately 6.5% is significantly larger than the grid uncertainty. Note that authors believe that the iterative uncertainty is caused by the innate unsteadiness of the flow at such a large length scales, rather than the iterative procedure for the non–linear equations sets. The viscous force obtained with the fine grid is approximately 137.071 kN, which is 2% below the ITTC 1957 correlation for the achieved speed. Figure 5a presents the dynamic pressure field at the bow, where a breaking bow wave can be seen due to vertical, cylindrical bow stem without a bulb. The dynamic pressure field at the stern and at the propeller plane represented by the actuator disc is shown in Figure 5b. Note that the dynamic pressure jump at the propeller plane is not symmetric due to the static roll angle as measured on sea trials [11], which is taken into account in present CFD simulations.

CPU times for all self propulsion simulations are summarised in Table 2. All simulations are carried out in parallel using up to 7 nodes (56 cores) on a distributed memory computational
Vuko Vukčević, Hrvoje Jasak, Inno Gatin, Tessa Uroić

Figure 4: Convergence of forces in CFD simulations.

(a) Convergence of resistance and thrust for fine grid,

(b) Convergence of resistance for the three grids.

Figure 5: Dynamic pressure field in bow and stern regions.

(a) Dynamic pressure at the bow,

(b) Dynamic pressure at the stern.

cluster: CPU–2x Intel Xeon E5–2637 v3 4–core, 3.5GHz, 15MB L3 Cache, DDR4–2133, with
InfiniBand communication. Convergence is achieved after 39.4 hours (1.6 days) wall clock time
for the coarse grid and after 83.8 hours (3.5 days) for the fine grid. It is interesting to note that
one second of real time for such a full scale simulation requires \( O(10^2) \) seconds of CPU time
using these computational resources. The main enabler for a large time step of \( \Delta t = 0.075 \) s is
the actuator disc model. If the discretised, rotating propeller is used, the time–step would be
approximately 100 times lower in order to resolve propeller motion with 0.5 degrees per time
step (e.g. as used by Shen et al. [14]).

4 CONCLUSION

This paper presents a direct comparison of ship scale CFD simulations with sea trial mea-
surements. A single case is simulated where the ship is free to sail given a constant propeller
revolution rate. The CFD simulations are performed with an actuator disc model, for which the
thrust and torque curves are generated with ship scale open water simulations using the MRF
approach. Three computational grids are used, ranging from 5 to 11 million of cells in order to
Table 2: CPU times for self propulsion simulations.

<table>
<thead>
<tr>
<th>Grid</th>
<th>Coarse</th>
<th>Medium</th>
<th>Fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of cells</td>
<td>5,597,931</td>
<td>7,469,642</td>
<td>11,727,781</td>
</tr>
<tr>
<td>Number of cores</td>
<td>48</td>
<td>48</td>
<td>56</td>
</tr>
<tr>
<td>CPU time per time-step, s</td>
<td>14.2</td>
<td>19.1</td>
<td>29.8</td>
</tr>
<tr>
<td>CPU time per second of real time</td>
<td>189.8</td>
<td>254.5</td>
<td>397.2</td>
</tr>
<tr>
<td>CPU time until convergence (t = 750 s), h</td>
<td>39.4</td>
<td>53.1</td>
<td>83.8</td>
</tr>
</tbody>
</table>

assess numerical uncertainty.

CFD results for the achieved ship speed and dynamic trim compare well with two sea trial measurements, where the grid refinement study reveals low numerical uncertainties. The discrepancy between successive CFD simulations using three grids is smaller than the difference between two sea trial measurements, which is expected since the weather and other conditions cannot be fully controlled during a sea trial. The successful comparison with sea trials indicates that the present CFD model with actuator disc for propeller can be readily used to estimate the ship speed, thus avoiding the need of CPU–time consuming discretised, rotating propeller.

Although the overall CPU time of approximately 7 days for three simulations can be considered reasonable for ship scale simulations, the future effort shall be focused on investigating the possibility of using smaller and coarser grids with more suitable refinement regions in order to further decrease the CPU time.

5 ACKNOWLEDGMENTS

Considering how difficult it is to find publicly available results from sea trials, we would like to thank and acknowledge Lloyd’s Register for organising and hosting the Workshop on ship scale hydrodynamic computer simulation [10] and making the valuable data from sea trials available, thus enabling us to perform a direct comparison of CFD with sea trials.

REFERENCES


SYSTEM-BASED MANEUVERING SIMULATION OF A SHIP NAVIGATING IN THE CONFINED WATERWAY

P. DU*, A. OUAHSINE*, P. SERGENT†

* Laboratoire Roberval, UMR-CNRS 7337
Sorbonne Universités, Université de Technology de Compiègne
Centre de Recherches Royallieu, CS 60319, 60203 Compiègne cedex, France
e-mail: pp1565156@126.com (P. Du); ouahsine@utc.fr (A. Ouahsine)

† CEREMA-134, rue de Beauvais, CS 60039, 60200 Compiègne, France

Key words: Maneuvering, confined waterway, bank effect, trajectory

Abstract. The system-based maneuvering simulations were conducted to investigate the ship navigating in the confined waterway. The confinement effect was included using the model of Vantorre [1]. The maneuvering model was validated using the turning circle and zigzag tests, and the confinement model was verified using the experimental data of Norrbin [2]. The good agreement proved the validity of our method. Using this method, the influences of the ship-bank distance and the propeller rate of turn were studied and concluded. Small ship-bank distance and large propulsion were proved to enhance the confinement effect.

1 INTRODUCTION

When a ship is maneuvered in a confined waterway, its motion will be influenced by the channel bank and bottom [3, 4, 5, 6]. These effects will change the maneuverability of the ship and may cause marine accidents if not controlled well. In this paper, system-based maneuvering simulations are conducted for an Esso Bernicia 190,000 DWT tanker in a confined waterway. A non-linear maneuvering model is adopted and the confinement effect is included based on the model of Vantorre [1]. The maneuvering model is verified using the turning circle and zigzag tests [7]. The confinement model is validated using the experimental data of Norrbin [2].

2 METHODOLOGIES

The maneuvering equations in 3-DOF (degrees of freedom) can be written as (Fig.1) [8]:
\begin{align*}
\dot{u} - vr - x_Gr^2 &= gX_n \\
\dot{v} + ur + x_Gr^2 &= gY_n \\
(Lk_z^n)^2 \dot{r} + x_G(\dot{v} + ur) &= gLN_n
\end{align*}

(1)

where \(u\) and \(v\) are the surge and sway velocities, \(\dot{u}\) and \(\dot{v}\) are the surge and sway accelerations, \(r\) and \(\dot{r}\) are the yaw rate and yaw acceleration respectively, \(g\) is the gravitational acceleration, \(G\) is
the gravity center of the ship, $O$ is the midship point, $L$ is the ship length between perpendiculars, $x_G$ is the position of the gravity center in OX-direction. $k_z^d = \frac{1}{L} \sqrt{\frac{T}{m}}$ is the non-dimensional radius of turning, where $I_z$ is the inertial moment of a ship with respect to OZ-axis. $X^u = X/(mg), Y^u = Y/(mg), N^u = N/(mgL)$ are the non-dimensional forces and moment.

For the Esso Bernicia 190,000 DWT tanker, the non-dimensional equations for surge, sway and yaw dynamics are:

$$
gX^u = X^u_{\dot{u}} + \frac{1}{L} X^u_{\dot{u}|u} u |u| + X^u_{uv} v r + \frac{1}{L} X^u_{\dot{v}|v} v |v| + \frac{1}{L^2} X^u_{\dot{\delta}|\delta} c |c| \delta^2 + \frac{1}{L} \sum_{\alpha=1}^{N_p} \delta \hat{c}(\alpha) \beta^\alpha + gT^u (1 - t_d) + \frac{1}{L} X^u_{\dot{\xi}} u \dot{\xi} + \frac{1}{L} X^u_{\dot{\xi}v} v \dot{\xi} + \frac{1}{L} X^u_{\dot{\xi}v^2} \dot{\xi}^2 \tag{2}
$$

$$
gY^u = Y^u_{\dot{v}} + \frac{1}{L} Y^u_{uv} u v + \frac{1}{L} \sum_{\alpha=1}^{N_p} \frac{1}{\alpha!} Y^u_{\dot{v}|\delta|\delta} c |c| \delta^\alpha + \frac{1}{L} Y^u_{\dot{\xi}|\xi|\xi} v |v| + \frac{1}{L} \sum_{\alpha=1}^{N_p} \frac{1}{\alpha!} Y^u_{\dot{\xi}|\delta|\delta} c |c| \delta^\alpha + gT^u + Y^u_{\dot{\xi}} ur \dot{\xi} + \frac{1}{L} Y^u_{\dot{\xi}v} u \dot{\xi} + \frac{1}{L} Y^u_{\dot{\xi}v^2} \dot{\xi} \tag{3}
$$

$$
gLN^u = L^2 N^u_{\dot{r}} + N^u_{\dot{r}r} + L^2 N^u_{\dot{\xi}|\xi} v |v| + \frac{1}{L} \sum_{\alpha=1}^{N_p} N^u_{\dot{\xi}|\delta|\delta} c |c| \delta^\alpha + L N^u_{\dot{r}r} \dot{\xi} + N^u_{\dot{\xi}|\xi|\xi} v |v| + \frac{1}{L} \sum_{\alpha=1}^{N_p} N^u_{\dot{\xi}|\delta|\delta} c |c| \delta^\alpha + gT^u + L N^u_{\dot{\xi}r} v \xi + L^2 N^u_{\dot{\xi}v} \xi \dot{\xi} \tag{4}
$$

where $X^u_{\dot{u}}, X^u_{\dot{u}|u}, ..., Y^u_{\dot{v}}, Y^u_{\dot{v}v}, ..., N^u_{\dot{\xi}}, N^u_{\dot{\xi}\xi}, ..., N^u_{\dot{\delta}|\delta}\delta$ are the non-dimensional derivatives of ship hydrodynamic coefficients. $\delta$ is the rudder angle, $t_d$ is the thrust deduction coefficient. $\beta = \arctan(-v/u)$ is the drift angle. $\xi = T_d/(h - T_d)$, where $h$ is the water depth, $T_d$ is the ship draft. $T^u$ is the non-dimensional propeller thrust given by:

$$
T^u = \frac{1}{g} T_{uu} u^2 + \frac{1}{g} T_{un} u n + \frac{1}{g} T_{n|n} n^2 \tag{5}
$$

where $T_{uu}, T_{un}$ and $T_{n|n}$ are the hydrodynamic coefficients of the propeller, $n$ is the shaft velocity. $c$ is the flow velocity at the rudder estimated by:

$$
c^2 = c_{uu} u^2 + c_{nn} n^2 \tag{6}
$$

where $c_{uu}$ and $c_{nn}$ are the hydrodynamic coefficients of the rudder.

The confinement effect is included as external forces and moments acted on the ship hull into the non-dimensional ship maneuvering equations. Then Eq.1 becomes:

$$
\dot{u} - v r - x_G r^2 = gX^u + gX^u_B \tag{7}
$$

$$
\dot{v} + u r + x_G r^2 = gY^u + gY^u_B
$$

$$
(Lk_z^d)^2 \dot{r} + x_G (\dot{v} + ur) = gLN^u + gLN^u_B
$$

where $X^u_B, Y^u_B$ and $N^u_B$ are non-dimensional forces and moment of the ship-bank/bottom interaction (Fig.2). According to [1], they can be decomposed as:

$$
X^u_B \approx 0
$$

$$
Y^u_B = Y^H_B + Y^P_B + Y^{HP}_B
$$

$$
N^u_B = N^H_B + N^P_B + N^{HP}_B \tag{8}
$$
Figure 1: Coordinate system in the ship maneuvering analysis

Figure 2: Parameter definitions of a ship in the confined waterway

where $Y_B^H$, $N_B^H$ are the effects of the forward speed, $Y_B^P$, $N_B^P$ are the effects of the ship propulsion, $Y_B^{HP}$, $N_B^{HP}$ are the coupled effects of the forward speed and propulsion.

\[
Y_B^H = \frac{1}{2} \rho L T_d u^2 \sum_{i=1}^{2} \sum_{k=0}^{2} \alpha_{ik}^H y_B^i \left( \frac{T_d}{h_{eff} - T_d} \right)^k
\]
(9)

\[
N_B^H = \frac{1}{2} \rho L^2 T_d u^2 \sum_{i=1}^{2} \sum_{k=0}^{2} \beta_{ik}^H y_B^i \left( \frac{T_d}{h_{eff} - T_d} \right)^k
\]
(10)

\[
Y_B^P = \frac{1}{2} \rho L T_d V_T^2 \sum_{i=1}^{2} \sum_{k=0}^{2} \alpha_{ik}^P y_B^3 \left( \frac{T_d}{h_{eff} - T_d} \right)^k
\]
(11)

\[
N_B^P = \frac{1}{2} \rho L^2 T_d V_T^2 \sum_{i=1}^{2} \sum_{k=0}^{2} \beta_{ik}^P y_B^3 \left( \frac{T_d}{h_{eff} - T_d} \right)^k
\]
(12)
Table 1: Geometrical parameters of the Esso Bernicia 190,000 DWT tanker

<table>
<thead>
<tr>
<th>Geometrical parameters</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>L</td>
<td>304.8 (m)</td>
</tr>
<tr>
<td>B</td>
<td>47.17 (m)</td>
</tr>
<tr>
<td>T_d</td>
<td>18.46 (m)</td>
</tr>
<tr>
<td>( \nabla ) (displacement)</td>
<td>220000 (m³)</td>
</tr>
<tr>
<td>L/B</td>
<td>6.46</td>
</tr>
<tr>
<td>B/T_d</td>
<td>2.56</td>
</tr>
<tr>
<td>C_B (block coefficient)</td>
<td>0.83</td>
</tr>
<tr>
<td>U_0 (design speed)</td>
<td>16 (knots)</td>
</tr>
<tr>
<td>n (propeller rate of turn)</td>
<td>80 (rpm)</td>
</tr>
</tbody>
</table>

\[ Y_{B}^{BP} = \frac{1}{2} \rho \frac{L}{T} \frac{V_{T}^{2} F_{r}}{T} \sum_{i=1}^{2} \sum_{k=0}^{2} \alpha_{ik}^{HP} y_{B3} \left( \frac{T_{d}}{h_{eff} - T_{d}} \right)^{k} \]

\[ N_{B}^{BP} = \frac{1}{2} \rho \frac{L}{T} \frac{V_{T}^{2} F_{r}}{T} \sum_{i=1}^{2} \sum_{k=0}^{2} \beta_{ik}^{HP} y_{B3} \left( \frac{T_{d}}{h_{eff} - T_{d}} \right)^{k} \]

where \( \alpha_{ik}^{H}, \beta_{ik}^{H}, \alpha_{ik}^{P}, \beta_{ik}^{P}, \alpha_{ik}^{HP}, \beta_{ik}^{HP} \) are the regression coefficients, which can refer to the work of Vantorre [1]. The reference velocity \( V_T \) is introduced as:

\[ V_T = \sqrt{\frac{T}{\frac{1}{8} \rho \pi D^2}} \]

where \( T \) is the propeller thrust, \( D \) is the propeller diameter, \( \rho \) is the water density. \( y_B \) and \( y_{B3} \) are non-dimensional quantities defined by:

\[ y_B = \frac{1}{2} B \left( \frac{1}{y_p} + \frac{1}{y_s} \right) \]

\[ y_{B3} = \frac{1}{2} B \left( \frac{1}{y_{p3}} + \frac{1}{y_{s3}} \right) \]

where \( y_p \) and \( y_{p3} \) are the distances from the ship center to the bank on the port side (Fig.2), \( y_s \) and \( y_{s3} \) are those on the starboard side, \( B \) is the breadth of the ship. \( h_{eff} \) is the effective depth of the channel:

\[ h_{eff} = h - z_m \]

where \( h \) is the water depth, \( z_m \) is the average sinkage due to the squat effect.

3 MANEUVERING MODEL TESTS

This work uses the Esso Bernicia 190,000 DWT tanker, whose parameters are given in Tab.1. There are 34 hydrodynamic coefficients in the maneuvering equations (2-4), whose values are given in Tab.2. Here the hydrodynamics are optimized using the procedures in [8].
Table 2: Hydrodynamic coefficients of the Esso Bernicia 190,000 DWT tanker

The maneuvering model here is validated using the turning circle and zigzag tests. As shown in Figs.(3-4), the trajectory and heading angle correspond well with the experimental data. Thereby this model can be further used to implement the ship-bank/bottom interaction in the confined waterway.

4 SHIP MANEUVERING IN A CONFINED WATERWAY

Based on the maneuvering equations 7, the ship navigating in the confined waterway can be simulated. The experimental results of Norrbin [2] are selected for validation. The setups can be seen in Tab.3. As shown in Fig.5, our simulations correspond well with the experimental data.

Two cases are further carried out to consider the influences of the ship-bank distance and the propeller rate of turn (Tab.3). As shown in Fig.6, when the ship is closer to the bank, the bank effect will become more obvious and the ship will deviate more from the bank. In Fig.7, when the propeller revolution increases, according to the equations 9-14, the forces and moment induced by the confinement effect will also increase. So greater propulsion will make the ship turn more to the middle of the channel. Overall, small ship-bank distance and large propeller rate of turn will enhance the confinement effect when a ship is maneuvered in the confine waterway.
Figure 3: Ship trajectories in the turning circle test. ’o’: experimental result [7]; ’-’: simulation result.

Figure 4: Yaw angles in the zigzag test. ’o’: experimental result [7]; ’-’: simulation result.
Figure 5: Trajectory of the ship navigating in the confined waterway (validation). ‘o’: experimental result [2]; ‘−−’: simulation result.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Validation</th>
<th>Case 1</th>
<th>Case 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>$L_b$ (width of channel bottom) [m]</td>
<td></td>
<td>300</td>
<td>300</td>
</tr>
<tr>
<td>$L_t$ (width of channel top) [m]</td>
<td></td>
<td>305</td>
<td>453</td>
</tr>
<tr>
<td>Channel slope of both sides</td>
<td></td>
<td>10:1</td>
<td>1:3</td>
</tr>
<tr>
<td>$h$ (water depth) [m]</td>
<td></td>
<td>25.5</td>
<td>25.5</td>
</tr>
<tr>
<td>$T_d$ (draft) [m]</td>
<td></td>
<td>18.46</td>
<td>18.46</td>
</tr>
<tr>
<td>$UKC$ (Under Keel Clearance) [m]</td>
<td></td>
<td>7.04</td>
<td>7.04</td>
</tr>
<tr>
<td>$y_{s3}$ (distance to the right bank) [m]</td>
<td></td>
<td>77.1</td>
<td>98.8,123.8,148.8,173.8</td>
</tr>
<tr>
<td>$u_0$ (initial ship speed) [knots]</td>
<td></td>
<td>5.0</td>
<td>5.0</td>
</tr>
<tr>
<td>$n$ (propeller rate of turn) [rpm]</td>
<td></td>
<td>80</td>
<td>80</td>
</tr>
</tbody>
</table>

Table 3: Simulation parameters in different cases
Figure 6: Ship trajectories with different ship-bank distances (case 1)

Figure 7: Ship trajectories with different propeller revolutions (case 2)

5 CONCLUSIONS

- The confinement model was successfully implemented into the maneuvering equations to simulate the ship navigating in the confined waterway.

- Small ship-bank distance and large propeller rate of turn can increase the confinement effect and influence the maneuvering of the ship.

REFERENCES


Towards CFD guidelines for planing hull simulations based on the Naples Systematic Series

Simone Mancini*, Fabio De Luca* and Anna Ramolini#

* Department of Industrial Engineering
Università degli Studi di Napoli “Federico II”
Via Claudio 21, 80125, Naples, Italy
e-mail: simone.mancini@unina.it, fabio.deluca@unina.it, web page: http://www.dii.unina.it

# TU Delft Faculty of Aerospace Engineering
Delft University of Technology
Kluyverweg 1, 2629 HS Delft, Netherlands
e-mail: a.ramolini@student.tudelft.nl, web page: http://www.lr.tudelft.nl/en/

ABSTRACT

Due to their higher motion amplitudes and instabilities, numerical simulations of planing hulls using Computational Fluid Dynamics (CFD) codes are more difficult than that of displacement ships. Indeed, for an accurate evaluation of the hydrodynamic performances of planing craft, the high-fidelity estimation of the pressure field around the hull is crucial. For this reason, validations and comparisons with experimental data are still important to identify the guidelines for both simulation settings and mesh generation. In this paper, two commercial packages will be compared focusing on a resistance case for the parent hull model (C1 hull) from the Naples Systematic Series (NSS) at four Froude numbers (Fr).

The NSS is a new systematic series of hard chine hulls intensively tested in planing and semiplaning speed range, De Luca et al. [1]. It has been chosen for the hull form: it is characterized by a warped bottom and a sectional area curve significantly different from the prismatic hulls. These differences amplify the difficulties in finding out the exact pressure distribution on the bottom and, consequently, make the evaluation more stringent.

The Unsteady Reynolds-Averaged Navier-Stokes flow solvers results are validated using these benchmark experimental data. Also, grid independence, iteration, and time-step convergence analysis for response variables (resistance coefficients, wetted surfaces, and dynamic trim angles) follow the recommendations published in the verification and validation (V&V) study from De Luca et al. [2]. Hence, the two software are more compared on different features such as the mesh deformation, the overset method, and the correction of numerical ventilation classically observed below the hull. The results show that both software can provide consistent values and that new guidelines are now identified to improve the reliability of the simulations.

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BWL</td>
<td>Water line breadth (m)</td>
</tr>
<tr>
<td>COG</td>
<td>Center Of Gravity (m)</td>
</tr>
<tr>
<td>E</td>
<td>Comparison error</td>
</tr>
<tr>
<td>D</td>
<td>Experimental data</td>
</tr>
<tr>
<td>Fr</td>
<td>Froude number</td>
</tr>
<tr>
<td>LNL</td>
<td>Water line length (m)</td>
</tr>
<tr>
<td>LPP</td>
<td>Length between perpendicular (m)</td>
</tr>
<tr>
<td>WK</td>
<td>Wetted keel length (m)</td>
</tr>
<tr>
<td>C</td>
<td>Total drag coefficient</td>
</tr>
<tr>
<td>CF</td>
<td>Frictional drag coefficient</td>
</tr>
<tr>
<td>FX</td>
<td>Longitudinal drag (N)</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>S</td>
<td>Wetted surface (m²)</td>
</tr>
<tr>
<td>SNS</td>
<td>Numerical Simulation result</td>
</tr>
<tr>
<td>V</td>
<td>Hull speed (m/s)</td>
</tr>
<tr>
<td>ΔD</td>
<td>Displacement (kg)</td>
</tr>
<tr>
<td>NSS</td>
<td>Naples Systematic Series</td>
</tr>
<tr>
<td>V&amp;V</td>
<td>Verification and Validation</td>
</tr>
</tbody>
</table>

1071
1. Introduction

It is known that simulating high speed flows around planing hulls is not a trivial task with the available CFD codes, and therefore several strategies were investigated in De Luca et al. [2]. This publication investigated the analysis of the flow around a planing bare hull model taken from the Naples Systematic Series. The results were compared against experimental data obtained in the towing tank at the Naval Division of the Dipartimento di Ingegneria Industriale of the Università degli Studi di Napoli “Federico II”. The available experimental results were provided at speeds ranging from 2.0 to 7.5 m/s, with an interval of 0.5 m/s, while the simulations were ran for four different speeds, namely 4.0, 5.0, 6.0 and 7.0 m/s. Two ways of approaching the problem were tested and finally recommended, mainly differing from the meshing strategy point of view, split into two categories:

1. “Single deforming mesh”, also called “weighted deformation”
2. “Overset mesh”, also called “overlapping grids” or “chimera approach”

From this analysis (De Luca et al. [2]), that tested the overset grid at different levels of coarseness other than the single mesh, it resulted that the coarsest case of the overset grid was the best compromise between accuracy and CPU time. In fact, the single mesh and finer overset yielded small improvements of the results but required around 400 s per timestep, against the 90 s per timestep of the coarsest overset case. These conclusions were obtained using STAR-CCM+ from Siemens.

The idea of this present publication is to try to replicate the conclusions using FINE/Marine from NUMECA International using both the single mesh deformation and the overset method. On top of the single mesh deformation, FINE/Marine also proposes a combination of the deforming mesh with Savitsky prediction. The idea is to be able to mesh the vessel in the final estimated position and speed up the simulation. Hence, this strategy differs with respect to the initial position of the boat, while the overset strategy requires a whole different mesh.

In all cases, the authors tried to replicate as much as possible the meshes used in CD Adapco STAR CCM+, in order to provide a fair comparison since there was no clear way to share the mesh between the two softwares. All approaches yielded satisfactory results, and they will be compared in a more detailed way in the following sections.

2. Test case description

The model used for the experimental testing is the parent hull of the Naples Systematic Series. The name of this model is C1. The other four models of the series were obtained by systematically scaling breadth and depth by the same reduction factors, without changing the transversal shape. The main feature of this series is that its hulls have a warped bottom and their sectional area differs from the one of the classic prismatic hulls. Figures from 1 to 4 show the C1 model in different views. In Table 1 all the relevant information about the model is reported.
Table 1. Model C1 hull data.

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>L&lt;sub&gt;WL&lt;/sub&gt; [m]</td>
<td>2.4</td>
</tr>
<tr>
<td>B&lt;sub&gt;WL&lt;/sub&gt; [m]</td>
<td>0.743</td>
</tr>
<tr>
<td>D [m]</td>
<td>0.46</td>
</tr>
<tr>
<td>Δ[kg]</td>
<td>106.7</td>
</tr>
<tr>
<td>S&lt;sub&gt;WS&lt;/sub&gt; [m&lt;sup&gt;2&lt;/sup&gt;]</td>
<td>1.7</td>
</tr>
<tr>
<td>L/B</td>
<td>3.23</td>
</tr>
<tr>
<td>x COG from bottom of transom [m]</td>
<td>0.943</td>
</tr>
<tr>
<td>z COG from bottom of transom [m]</td>
<td>0.193</td>
</tr>
<tr>
<td>Deadrise x/L=0 [deg]</td>
<td>13.2</td>
</tr>
<tr>
<td>Deadrise x/L=0.5 [deg]</td>
<td>22.3</td>
</tr>
<tr>
<td>Deadrise x/L=0.75 [deg]</td>
<td>38.5</td>
</tr>
</tbody>
</table>

Given the symmetry of the model with respect to the Y axis (y COG=0) the simulations have been performed on half of the ship in order to decrease the computational time required.

3. Meshing strategy

This section describes the meshing strategy, created using the unstructured NUMECA’s mesh generator HEXPRESS.
3.1. Single mesh

The single deforming mesh is built using one single domain, its size is determined following the recommendations as shown in Figure 5:

Table 2 compares the single mesh base size and total number of cells of the FINE/Marine and STAR CCM+ meshes.

<table>
<thead>
<tr>
<th></th>
<th>FINE/Marine</th>
<th>STAR CCM+</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base size [m]</td>
<td>0.133</td>
<td>0.1128</td>
</tr>
<tr>
<td>Total number of cells [-]</td>
<td>1 703 724</td>
<td>1 855 777</td>
</tr>
</tbody>
</table>

A first box refinement is defined around the boat, with target cell size 0.039 m in all directions. This box is shown with a dash blue line in Figure 6.
The undersea refinement can also be seen in Figure 6 with a dash orange line as a smaller box located at the bottom of the hull: this box will have one extra refinement compared to the other box and therefore its target cell size will be 0.0196 m. Thanks to this box it will be easier to capture the interaction between the free surface and the hull.

The free surface will also need to be refined, especially in the z direction, to capture the air-water interface. For this purpose an internal surface at the free surface level will be created, and the refinement will be applied only in the z-direction, with target cell size 0.0196 m. On the same internal surface more refinements are needed in the x and y direction. The wake is then refined by creating lofted surfaces (Figure 7). The target cell size in x and y direction will be also 0.0196 m.

![Figure 7. Wake refinement.](image)

For all the solid surfaces a target cell size of 0.0165 m has been chosen, while the curves have one extra level of refinement, with target cell size 0.011 m. This is done to ensure that the shape of the boat is correctly captured even in the most difficult areas, e.g. at the bow where 5 hull lines converge in the same point. The meshed half boat is shown in Figure 8.

Once all refinements are defined, it is necessary to add viscous layers in correspondence of all the solid patches (except for the deck) to accurately resolve the boundary layers. 11 layers will be applied to the hull and the transom.

![Figure 8. Meshed half boat.](image)
3.2. Overset mesh

The overset mesh creation is a bit more challenging than the one of the single mesh, since it requires the creation of two computational domains: the background and the overlapping domain. The first one consists in a box of large dimensions, while the second one is smaller and contains the boat. The idea underlying the overset technique is that the overlapping mesh rigidly follows all the motions of the ship, while the background domain just follow the forward motion of the boat. The advantage of this technique is that there no mesh deformation ensuring the optimum mesh quality all along the simulation. On the other hand, the drawback lays in the fact that there has to be communication between the two domains, through an interpolation across the domain boundaries of the smaller domain. In order to limit the effect of the interpolation to the minimum the meshes have to be built so that the size of the cells at the boundary of the overlapping domain is as close as possible to the one of the background domain in that area.

First of all, the domain sizes are defined following what is shown in De Luca et al. [2], reported in Figure 10 for clarity. Once the domains are created, the same refinements presented for the single mesh are applied, with the only difference that some refinements will belong to the overset domain and some to the background. The free surface refinement will belong to both: an internal surface needs to be created in both domains. The wake refinement will be present only in the background domain, while the surface, curve and undersea refinements will be added in the overset one. Another refinement is added in the background domain, to comply with the requirement of same cell size in the area around the boundaries of the overset domain. This is done by inserting in the background domain a box refinement around the overset domain with target cell size 0.04 m. Figure 11 shows how the requirement is well met throughout the boundary, having a 1-to-1 ratio between the cell sizes of the two domains across the interface.

![Figure 10: Background and overset domain sizes.](image-url)
Table 3 and Figure 12 show the comparison between the mesh generated by HEXPRESS and STAR CCM+. As there was no easy way to share the mesh between the two softwares, the size of the refinement areas was not known precisely and had to be estimated, and this led to a variation in the total number of cells. Another factor that might have contributed to have different number of cells is that HEXPRESS uses a refinement diffusion that can also be anisotropic, as it can be observed in the top right of Figure 7, while it is not the case for STAR CCM+.

Table 3. Base cell size and total number of cells of the meshes generated with the two softwares.

<table>
<thead>
<tr>
<th></th>
<th>FINE/Marine</th>
<th>STAR CCM+</th>
</tr>
</thead>
<tbody>
<tr>
<td>Overset base size [m]</td>
<td>0.592</td>
<td>0.9</td>
</tr>
<tr>
<td>Background base size [m]</td>
<td>2.476</td>
<td>2.299</td>
</tr>
<tr>
<td>Total number of cells [-]</td>
<td>1 196 583</td>
<td>813417</td>
</tr>
</tbody>
</table>
3.3. Savitsky mesh

This mesh is equivalent to the deforming single mesh, the only difference is the initial position of the boat: while in first approach, the vessel was aligned with the $x$-axis, it is now translated and rotated around its center of gravity of an angle computed with the Savitsky estimation. This method, explained in detail in Savitsky [3], allows to estimate the final trim, sink, lift, drag and wetted area of prismatic planing hulls given some initial data such as speed, center of gravity, deadrise angle and mass. The more the hull is shaped like a triangle, the more accurate this estimation becomes. In the case treated in this paper the hull is warped and therefore the estimation will not be totally accurate. The advantage of this technique is that the computation can start already at the final speed, and since the initial position of the boat should be closer to the final one, convergence should be reached faster.

The Savitsky estimation method embedded into FINE/Marine requires a different mesh for each Froude number, since a different speed implies a different initial trim and sink. Therefore, 4 meshes were created for this case. Some adjustments were then made to have the same type of refinements as the other two meshes. The total number of cells in this case is around 1 700 000, similarly to the single mesh case.

4. Project settings

In this section the parameters chosen for the simulations are shown and explained. In Table 4 the fluid properties are summarized. The chosen time configuration is steady, since the flow surrounding the vessel is uniform and constant in time. The turbulence model used is $k$-$\omega$ SST, which is generally the standard one for marine applications. In De Luca et al. [2] the $k$-$\varepsilon$ model was used, but the authors also showed in this paper that the differences in the results obtained with the two models were negligible (Figure 13).
Figure 13: Results obtained with two different turbulence models with STAR CCM+.

Table 4: Fluid properties.

<table>
<thead>
<tr>
<th></th>
<th>Density [kg/m³]</th>
<th>Dynamic viscosity [Pa s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water</td>
<td>997.56</td>
<td>0.00104</td>
</tr>
<tr>
<td>Air</td>
<td>1.205</td>
<td>$1.81 \times 10^{-5}$</td>
</tr>
</tbody>
</table>

The applied boundary conditions are summarized in Table 5.

Table 5: Applied boundary conditions.

<table>
<thead>
<tr>
<th>Patch</th>
<th>BC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solid patches except Deck</td>
<td>Wall function</td>
</tr>
<tr>
<td>Deck</td>
<td>Slip (zero shear stress)</td>
</tr>
<tr>
<td>Inlet</td>
<td>Far field, Vx=0</td>
</tr>
<tr>
<td>Outlet</td>
<td>Far field, Vx=0</td>
</tr>
<tr>
<td>Top</td>
<td>Updated hydrostatic pressure</td>
</tr>
<tr>
<td>Bottom</td>
<td>Updated hydrostatic pressure</td>
</tr>
<tr>
<td>Side 1 (y=0)</td>
<td>Mirror</td>
</tr>
<tr>
<td>Side 2</td>
<td>Far field, Vx=0</td>
</tr>
<tr>
<td>All overset boundaries except mirror plane</td>
<td>Overset</td>
</tr>
</tbody>
</table>
Concerning the body motion in the single and overset case, the trim and sink motions are set to “solved” while the translation in $x$ is imposed, with a half sinusoidal ramp motion law that brings the vessel from zero velocity to the desired speed in 200 timesteps. As it has been mentioned before, in the Savitsky case it is possible to start the computation with the vessel already at the desired speed, thus with no ramp. Therefore in this case the translation in $x$ will be imposed at constant speed, and the Cardan angles for motion reference axis will be initialized with the value predicted by Savitsky. Since the ship is moving, the initial velocity of the flow is set to zero in all cases. An additional wave damping model is added to better reproduce the experimental data, since a damper was present in the towing tank, and also to reduce the oscillations in the results. This damping model is the one introduced by Choi and Yoon (see ref. [4]), and the damping areas are the ones illustrated in Figure 14.

![Figure 14: Damping areas.](image)

The choice of the timestep was done following FINE/Marine recommendations and what has been done in De Luca et al. [2], namely:

$$\Delta t = 0.005l/V$$

where $V$ is the hull speed and $l$ is the $L_{WL}$. All computations were run until convergence, which means that a different number of time steps has been applied to each case with the only common requirement that the oscillations should be less than 1.0% of the final computed value.
5. Results and discussion

5.1 Comparison of results

In this section the results are presented in terms of percentage difference with respect to the experimental data. The difference is expressed as:

\[ D = \frac{E - S}{E} \]

Where \( D \) is the difference, \( E \) the experimental value and \( S \) the value computed in the simulation. Figures 15 and 16 show the absolute values of the differences obtained with the various methods and

![Figure 15](image1.png)

*Figure 15: Percentage difference between CFD simulation and EFD for the different methods used. Ct (left) and Cf (right)*

![Figure 16](image2.png)

*Figure 16: Percentage difference between CFD simulation and EFD for the different methods used for Sw (left), and CFD and EFD trim values (right)*
the two different softwares. The analyzed quantities are the resistance coefficient \( C_t \), the frictional resistance coefficient \( C_f \), the wetted surface \( S_w \) and the trim angle \( R_y \). The two coefficients are computed respectively as:

\[
C_t = \frac{F_x}{\frac{1}{2} \rho S_w V^2} \quad C_f = \frac{0.075}{(\log_{10} Re - 2)^2}
\]

Where \( F_x \) is the longitudinal force and \( Re \) is the Reynolds number based on the wetted keel length \( L_K \), as defined in De Luca et al. [2]. Regarding the results obtained with FINE/Marine, all computational results agree well with the experimental data, as shown in Fig. 15 and 16. For all the mesh configurations that were tested, the difference between the experimental and the CFD data is lower for the frictional resistance coefficient than for the other parameters. The difference for this quantity is particularly low because this parameter only depends on the wetted keel length \( L_K \), which is in almost perfect agreement with the experimental data in all computations. From the results it is possible to argue that the Savitsky approach gives the most satisfactory results, yielding the lowest differences in all computed quantities. As a general consideration it can be stated that even in the worst cases the differences are of totally acceptable entities for all the quantities: the highest difference percentages are found in the trim angle, but in these cases the difference in terms of degrees is never higher than 1 degree.

When comparing the results between the two softwares some observations can be made: FINE/Marine is reliable in computing the frictional coefficient (and therefore the wetted keel length), while STAR CCM+ yields better results in the estimation of the trim angle. Regarding the other analyzed quantities, the graphs show that the differences depending on the Froude number: for example, at low Froude it is preferable to use STAR CCM+ for the resistance coefficient, while FINE/Marine is more accurate at high Froude. The opposite can be stated for the wetted surface, where the overset mesh from STAR CCM+ gives the best estimate at high Froude while FINE/Marine is preferable for low Froude numbers.

5.2 Analysis of Savitsky prediction effects

In this section the Savitsky prediction will be analyzed and its effects on the computation will be shown. First of all it is worth mentioning the tool through which this estimation is performed: the C-Wizard. This plug-in for FINE/Marine is made to simplify the mesh and computation setup. It is made of Python scripts that automatically call existing macros and finish setup procedure with the minimal user input required. It has four main applications: resistance, seakeeping, open water and planing regime. In this case the planing regime will be chosen. This tool allows to import the selected geometry, input the necessary data for the computation and obtain the final trim and sink of the boat as an output using the estimation method from Savistsky.

The estimation method consists in an iterative process for which a trim angle is initially guessed and plugged in simplified equilibrium equations, at the end of each cycle a check is made and the process is stopped when equilibrium is reached (given a certain tolerance). The approximation in this method lays in the fact that the simplified equations are based only on certain types of hulls (prismatic planing hulls), and therefore the estimation will be more or less realistic the more or less the shape of the hull is close to the prismatic one.

The aim of this technique is to reduce the computation time since the estimation places the boat in the estimated final position. Figures 17 and 18 compare the convergence histories of trim and drag with and without Savitsky estimation for 4.0 m/s.

From the graphs it can be argued that the Savitsky estimation guarantees a faster convergence to the same value for both quantities, as it is shown by the vertical lines. In the Savitsky case the drag force stabilizes to its final value after around 4 seconds of physical time of simulation, while in the single mesh case the value can be considered converged only after 6 seconds, as it can be noted in the close up of Figure 18. For the trim angle the difference is even more appreciable: with Savitsky convergence
is reached after less than 4 seconds of physical time, while without the estimation only after around 6 seconds. The great improvement is due to the fact that the trim angle is initialized to a value that is closer to the final one, and therefore the latter is reached more quickly. Hence the Savitsky computation could have been stopped at 4 seconds of physical time, saving up to 33% CPU time.

![Figure 17: convergence history of trim angle at 4 m/s with and without Savitsky](image)

![Figure 18: convergence history of drag force at 4.0 m/s with and without Savitsky (left) and close up (right)](image)

5.3 Analysis of streaking correction effects

As it has been mentioned before, the computations on planing hulls are numerically more complicated than the regular ones, therefore the mass fraction can often result unphysical through a phenomenon called streaking, for which spray appears in areas where water should be present, as indicated by Ferziger and Peric [5] and by Andrillion and Alessandrini [6]. In order to avoid this source of error two numerical corrections have been implemented in FINE/Marine. These corrections are activated only where the viscous layers are present (hull and transom). In Figure 19 and 20 the difference in the mass fraction obtained with and without the corrections for the 4.0 m/s case can be appreciated.
The figures show how the correction is working correctly and is improving the solution guaranteeing a physically meaningful mass fraction distribution underneath the ship hull. Similar results were obtained with STAR-CCM+. Hence, it seems that these numerical corrections are necessary in the guidelines for high speed boat simulations.

6. Conclusions

A hydrodynamic investigation on the C1 model of the Naples Systematic Series has been done. Two commercial CFD softwares and various mesh strategies were compared. For both softwares the results show good agreement with the benchmark experimental data obtained at the University of Naples Towing Tank, but it can be stated that there is still room for improvement since most of the percentage differences are in the range 0-20%. This result can be considered anyways satisfactory given the history of CFD computations on planing hulls, that shows how are they much more difficult to perform correctly than the regular ones, as shown in De Luca et al. [2]. Inside the computations carried out with the FINE/Marine software, investigations have been done to evaluate the usefulness of some tools such as the Savitsky estimation and the streaking correction. Regarding the former, it has been shown that this tool can significantly speed up the computation and allow to reduce the simulation time up to 33%. The streaking correction showed to help limit the streaking phenomenon, typical of high speed simulations, for which the mass fraction is not physical in some areas due to numerical ventilation beneath the hull. The correction allows to ensure the physical meaningfulness of the mass fraction and therefore a correct estimation of the efforts on the hull. Overall it can be stated that CFD is definitely a useful tool when combined with experimental data to have reliable information on the behavior of a high speed planing boat.

7. Recommendations

The results presented in this paper show that commercial CFD softwares can estimate quantities like trim angle, wetted surface and resistance coefficient. For this particular test case various combinations of software and mesh strategies were investigated and it can be argued that some combinations are particularly accurate for the computation of some quantities while not so much for other ones. For example, the combination FINE/Marine-Overset mesh yields the most accurate estimation of the frictional coefficient, while the combination STAR-Overset mesh is preferable for estimation of trim angle. Therefore, general guidelines should still be investigated but trends seems to already appear. The users can use this paper as a first guideline to decide what strategy to use depending on the quantity they want to focus on in their analysis.
REFERENCES


A SCALABLE AND PRISMATIC PRESSURE VESSEL FOR
TRANSPORT AND STORAGE OF NATURAL GAS

PÅL G. BERGAN* AND DAEJUN CHANG†

* LATTICE Technology and Department of Structural Engineering
Norwegian University of Science and Technology (NTNU)
Richard Birkelands vei 1A, 7491 Trondheim, Norway
e-mail: pal.bergan@ntnu.no, web page: http://www.ntnu.edu/kt

† LATTICE Technology and Department of Mechanical Engineering
Korean Institute of Science and Technology (KAIST)
Yuseong-gu, Daejeon-si 34141, Republic of Korea
e-mail: daejun.chang@lattice-technology.com - Web page: http://www.LATTICE-Technology.com

Key words: Computational Methods, Marine Engineering, LNG, Pressure Vessel, Fuel Tank

Abstract. This paper deals with a new, innovative concept for storage and transportation of
gas, notably natural gas under cryogenic, liquefied, and pressurized conditions. This new
prismatic pressure vessel design differs largely from traditional pressure vessels that typically
are cylindrical or spherical shell structures. The current design principle is based on balancing
the pressure on opposite outer walls by way of an internal, force transferring tension structure.
This internal structure has the appearance of a lattice; thus, this type of tank is termed the
“Lattice Pressure Vessel” (LPV). The LPV is fully modular and scalable in the three spatial
directions. It has the potential of becoming a key component to facilitate the transition from
heavy fuel oil and marine fuel oil to the much cleaner natural gas and hydrogen as fuel for
propulsion of ships. The paper describes design principles, outlines how computational
analyses have been verified by comparing results with four different, instrumented test tanks.
Moreover, a series of examples of pressure vessel designs are given which illustrate the
benefits of structural efficiency as well as the ability of fitting such pressure vessels within the
limited space on board ships.

1 INTRODUCTION

The ambitious targets for sustainability and climate gas emissions set by the UN climate
conference in Paris in 2015 must be followed up with concrete actions of regulations and
practical measures of implementation. One example of such new regulations is the recently
approved, strict emissions requirements from ships to be implemented from year 2020.
Further, every country must act to reduce own emission of pollutants and green-house gases
with concrete targets for year 2030. This means that renewable energy should replace dirty
coal and oil fired energy production. Use of natural gas, rather than coal and oil, will be a key
step towards using cleaner fuels in energy production and transportation. To this end a new
global infrastructure for distribution, transport and storage of liquid natural gas (LNG) should
be built within few years.
The key element in distribution and use of LNG is the storage tanks. Large amounts of LNG, at the atmospheric boiling point of -163 degrees C, can be transported with ships with large insulated, unpressurized containments from liquefaction plants to major distribution facilities on land or at sea. However, the gas itself may largely be used in smaller power production facilities or for transportation purposes where the fuel tanks are relatively small and, consequently, heat ingress will lead to increased gas boil-off. This problem can be dealt with by complicated and expensive regasification systems or simply by containing the LNG within tanks that allow for pressure build-up (emission of methane-rich gas to the atmosphere is not allowed). The latter solution is clearly preferably in most cases.

The paper describes a “first of its kind” type of pressure vessel which is fully scalable in size and prismatic in shape rather than cylindrical or spherical. The design principle is based on balancing the pressure on opposite outer walls by way of an internal, force transferring tension structure. This internal structure has the appearance of a lattice; thus, this type of tank is termed the “Lattice Pressure Vessel” (LPV). The LPV is fully modular and scalable in all three spatial directions. Unlike cylinders, the thickness of the outer shell/gas barrier does not increase with tank size; the stiffened plate structure only depends on the design pressure. The concept has several other advantages over current pressure tank design such as ease of thermal insulation, overall space utilization efficiency, a redundancy-based safety concept, and ease of fabrication with moderate plate thicknesses and flat panels.

2 THE IDEA AND DESIGN CONCEPT

2.1 Classification of LNG tanks

The classification of containments for LNG is set down by IMO and defined as independent tanks or integrated tanks. The independent tanks are again divided between Type A with low pressure and full secondary barrier, Type B with low pressure and partial secondary barrier, and Type C which are pressurized tanks with no requirement for secondary barrier. Further, the integrated tanks are typically referred to as membrane tanks, which are low pressure tanks and rely on the ship hull for providing strength. Main codes that apply are the International Gas Code (IGC) [1], and the International Fuel Gas Code (IFC) for gas fuel tanks [2], and ASME’s boiler and pressure vessel code [3]. On top of this, any LNG tank must also comply with requirements set by the different classification societies.

The new pressure vessel described herein belong to the IMO Type C; however, IMO describes a pressure vessel as being cylindrical or spherical simply because prismatic pressure vessels have not been a consideration until now. In agreement with classification societies such as DNVGL the LPV is described as “Type C equivalent”. Clearly, all requirements in the present codes also apply for Type C equivalent tanks.

2.2 Design concept

The basic concept of the LPV is that pressure on the gas barrier shall mainly be carried by an internal load carrying structure rather than by the gas barrier itself as for cylindrical and spherical shell structures. An exemplification of this principle is shown in Figure 1 where the pressure on opposite walls of a prismatic pressure vessel is balanced by way of a series of parallel panels that the combine the two. A panel provides connection between both
horizontal surface panels and between vertical surface panels; this implies that the panel obtains a very efficient load and stress utilization. Clearly, the end plates require special attention since they are not directly supported by parallel panels. For large end plates this problem is dealt with by a special “end box design” which efficiently resolves this problem.

![Figure 1: Internal load carrying structure with parallel panels](image1)

Figure 1: Internal load carrying structure with parallel panels

Figure 2 illustrates the LPV principle in some further detail. Stiffening is required for the parallel panels for reasons of lateral strength and stability. Further, the outer skin is supported with stiffeners between the parallel panels to withstand the internal design pressure. All corners are rounded to minimize stress concentrations that in fact would be prohibitive for pressure vessels with sharp corners. All in all, the design is based on carefully balancing stiffness and deformations in such a way that the LPV deforms uniformly and the stress utilization is optimized without generating undesirable “hot-spots”.

![Figure 2: LPV with panels, skin stiffeners and rounded corners](image2)

Figure 2: LPV with panels, skin stiffeners and rounded corners

The internal structural system outlined in Figure 2 is not the only alternative that has been explored for LPVs. For instance, reference [4] deals with an internal structure that is composed of so-called X-beams. Normally the parallel panel design is more efficient than the X-beam design dealt with in the reference. Several patents have been obtained for the lattice designs, see e.g. [5] and [6].
3 ANALYSIS AND ENGINEERING

3.1 Design loads

The problem of designing partly liquid filled pressure vessels under cryogenic conditions onboard a moving ship in the ocean may be further understood when considering the number of load conditions that such tanks must be designed for. Table 1 shows an overview of such load cases that must be analyzed and designed for.

Table 1: Load cases that must be analyzed and designed for

<table>
<thead>
<tr>
<th>Load type</th>
<th>Loads</th>
<th>Analysis reports</th>
</tr>
</thead>
<tbody>
<tr>
<td>Permanent</td>
<td>Gravity, External</td>
<td>Structural analysis of LPV under dead load</td>
</tr>
<tr>
<td></td>
<td>Internal pressure, test</td>
<td>Structural analysis of LPV under internal pressure</td>
</tr>
<tr>
<td></td>
<td>Static heel</td>
<td>Structural analysis of LPV under static heel loads</td>
</tr>
<tr>
<td></td>
<td>Cargo weight</td>
<td>Structural analysis of LPV under cargo weight</td>
</tr>
<tr>
<td></td>
<td>External pressure</td>
<td>Structural analysis of LPV under external pressure</td>
</tr>
<tr>
<td>Functional</td>
<td>Thermal</td>
<td>Temperature distribution of LPV</td>
</tr>
<tr>
<td></td>
<td>Vibration</td>
<td>Modal analysis of LPV</td>
</tr>
<tr>
<td></td>
<td>Construction and installation</td>
<td>Stress analysis of lifting components</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Stress analysis of supporting structures</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Estimation of support load</td>
</tr>
<tr>
<td>Environmental</td>
<td>Ship motion</td>
<td>Estimation of liquid pressure load</td>
</tr>
<tr>
<td></td>
<td>Sloshing</td>
<td>Estimation of sloshing load</td>
</tr>
<tr>
<td>Accidental</td>
<td>Collision, flooding</td>
<td>Impact analysis of LPV</td>
</tr>
<tr>
<td>Combined</td>
<td>Combined loads</td>
<td>Structural analysis of LPV under combined static loads</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Structural analysis of LPV under combined dynamic loads</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Structural analysis of complex loading cases for LPV</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fatigue and fracture analysis of LPV</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Buckling analysis of LPV</td>
</tr>
</tbody>
</table>

It is apparent from this table that almost all branches of computational mechanics must be resorted to fully provide the required documentation of acceptable performance. Not only will the tank have to be analyzed with respect to stress and deformations, typically by way of finite element simulations, but also additional analyses will include heat conduction and thermal deformations, fluid phase transitions, dynamics, cracking and fatigue, stability and nonlinear failure analyses, fluid-structure interactions (sloshing), and even interaction between the ship and the LPV. Fortunately, current state-of-the-art of computational mechanics makes such analyses possible and good computational tools are available for this purpose. It will hopefully be understood that it will not be possible here to go into details about how these analyses are done in practice, however, the tasks themselves may be said to be interesting and challenging. It is also to be noted that the classification societies require full documentation of these aspects with reports more than 20 in number for the specific tank application considered. It may be of interest that the LPV design has an advantage over traditional shell or membrane tank solutions in that the LPV has internal structure that very efficiently dampens the fluid motion and the dynamic fluid pressure are small and without consequence. Still, it must be documented that sloshing is no problem for each case.
Figure 3 shows surface stresses for a pressurized LPV. Note the remarkably smooth and repetitive stress distribution patterns. The corners are clearly less stressed than the rest of the outer skin. Maximum stresses are noted along the midst of the plate fields next to the corners where the outer surface fibers are under compression. The codes allow higher stresses in the case where there is bending through the thickness rather than for uniform tension through the surface skin (as in a cylindrical shell).

3.2 Supports and interaction with the ship

For LPVs onboard a ship a consideration is to establish about how the LPV interacts with the ship itself. Clearly, this problem depends not only on the LPV, but just as much on the motion and deformation of the ship hull and the closest structural elements in the ship. Further, the pressure vessel will contract considerably due to the cooling to down to cryogenic conditions, such as down from ambient temperature to –163 deg C for LNG; such contraction may be as much as 10 to 20 cm for a large tank. Thermal deformations and stresses are function of temperature change in relation to ambient temperature and the thermal expansion (contraction) properties of the material. For the case where the entire tank is subject to uniform cooling there will be no thermal stresses generated, however, the support conditions must be allowing for uniform contraction.

Figure 4 shows an example of support system for a prismatic LPV tank. There are essentially two types of support, one that allows for constrained sliding in a prescribed, horizontal direction, and another that allows for free horizontal sliding and gives only vertical support. The support blocks are made of compressed wood that provides necessary insulation between the cold tank and the ship. Tracks in the wood blocks and steel guides provide necessary constraints for that force the sliding to be confined to the prescribed direction. The concept requires that one point is selected as fixed reference points and that all sliding supports should allow for motion in the direction from the support towards this fixed point. The function of the guides is to keep the tank in place and to be able to absorb gravity and inertia forces during ship motion.
The tank has also to be checked for thermal stressing during certain filling conditions in which there may be significant temperature gradient within the tank. The uniform structural system of the tank gives relatively small thermal stresses during such conditions. Moreover, an unventilated LNG tank is also subject to internal pressure build-up due to heat ingress; typical pressures to be designed for are 3 to 10 barg (0.3 to 1 MPa). The uniformity of the internal load bearing system leads a rather uniform expansion of the tank which is nicely dealt with the support system described before.

3.3 Thermal insulation

There are mainly two types of insulation used for cryogenic pressure vessels: polyurethane foam (PUF) and vacuum insulation. In the former case layers of PUF are sprayed onto the external surface of the tank, normally laid to thickness of 30 to 40 cm. Vacuum insulation is generally only used for relatively small tanks. In such case the insulation is provided by vacuuming a layer of perlite that is filled between the outer surface of the tank and an air tight plated external frame structure. An advantage by vacuum insulation is that the insulation layer only needs to be about 10 cm thick which means that there is more room for the tank itself within a given installation space in the ship. The heat ingress can be calculated along with determining the balance between liquid and vapor, and on this basis the holding time before reaching maximum allowed pressure can be accurately calculated.
4 TEST TANKS

4.1 Testing of four LPVs

Although the state of the art in computational mechanics is highly advanced, it has been necessary to carry out physical tests and demonstrate the performance and safety of the new LPV concept to classification societies as well as to the market at large. Four different test tanks have been built for this purpose with various volumes, shapes, materials and design pressure. Pressure testing has been carried out with pressurized water whereas cryogenic testing has been carried out using liquid nitrogen at –196 °C. All tests have been extensively instrumented with monitoring of pressure, deformations and strain. In all cases the maximum applied test pressure was 1.5 times the design pressure.

<table>
<thead>
<tr>
<th>Prototype tank</th>
<th>I</th>
<th>II</th>
<th>III</th>
<th>IV</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design pressure, barg</td>
<td>9.5</td>
<td>10.0</td>
<td>10.0</td>
<td>5.0</td>
</tr>
<tr>
<td>Hydraulic test pressure, barg</td>
<td>15.0</td>
<td>15.0</td>
<td>15.0</td>
<td>7.5</td>
</tr>
<tr>
<td>Dimension, H (m) x W (m) x L (m)</td>
<td>4 x 4 x 5</td>
<td>2.2 x 2 x 11.8</td>
<td>1.8 x 3.6 x 3.6</td>
<td>1.8 x 3.6 x 3.6</td>
</tr>
<tr>
<td>Volume, m³</td>
<td>80</td>
<td>50</td>
<td>22</td>
<td>22</td>
</tr>
<tr>
<td>Material</td>
<td>SA-516</td>
<td>High Manganese</td>
<td>SA-240</td>
<td>High Manganese</td>
</tr>
<tr>
<td>Target fluid</td>
<td>Water for test</td>
<td>LNG</td>
<td>LNG</td>
<td>LNG</td>
</tr>
<tr>
<td>Certificate</td>
<td>ASME U2</td>
<td>Design approval from KR</td>
<td>ASME U2 (Consent Letter from ABSC)</td>
<td>Design approval from KR</td>
</tr>
</tbody>
</table>

The results monitored for all test cases were fully in line with prior computational analyses and were thus highly satisfactory. Measured stresses, “hot-spots” and deformations were fully consistent with the numerical analyses and within the code requirements. The first tank with the design pressure 9.5 barg was in fact tested up to more than 20 barg (2.0 times the design pressure) without observing any indication of weakening. Extensive nonlinear finite analysis revealed that actual failure load would be more than 40 barg for this case. Further nonlinear simulations of various tanks confirmed that the design principles and structural layout of the LPV is soundly safe due to exceptional ability for the tank to redistribute forces due to structural topology and redundancy. In fact, the outer skin, which is the gas barrier, is the least
stressed part of a LPV. This property is quite different from cylindrical and spherical in which the gas barrier is also the only load bearing structural component; a weakening or crack in this barrier can easily lead to total tank failure.

4.2 Materials and fabrication

Pressurized cryogenic tanks require special materials while normal carbon steels cannot be used. The reason for this is strict requirement pertaining to strength, toughness and ductility at low temperature. The most used materials for this purpose are stainless steel, nickel steels and aluminum. POSCO has developed a new type of cryogenic steel, called high manganese steel, that is significantly cheaper than stainless and nickel steels. Two of the test tanks (No II and No IV) were built and tested using this material. The results obtained, including testing with liquid nitrogen, were very good.

Building of the test tanks also served the purpose of gaining experience with fabrication of LPVs; this all proved to be very satisfactory. In many ways, it is simpler to manufacture a prismatic tank consisting of stiffened plates of moderate thickness as compared with building cylindrical tanks with rather thick, curved plates for which the requirements for tolerance and precision are very strict. In fact, manufacturing of LPVs may be said to be quite similar to making a plated ship structure; the LPV tanks thereby lends themselves to be made with well established, automated fabrication methods.

5 EXAMPLES OF DESIGN AND APPLICATIONS

5.1 Rounded corners

A main parameter in selecting the type of pressure vessel to be used in a ship is the volume efficiency. By this factor is simply meant the ratio between the volume of the tank compared with the volume of the installation space in which the tank, or multiple tanks, are placed. It has been experienced that it is not possible to design a prismatic pressure vessel with sharp corners; the reason for this is simply that stress concentrations arising in such corners turn out to be unacceptably high. This problem is dealt with by rounding the corners and providing design details that are not conducive to generating stress concentrations. As illustrated in the pictures in Figure 6, rounded corners may be made with smaller or larger radius of curvature. An option for relatively “flat” tanks is that the corner radius is selected as half of the tank height, thereby replacing the entire set of side walls with semi-cylindrical, rounded walls. This turns out to be structurally efficient and gives lighter tanks, whereas, as shown in the associated table, this is at cost of a slightly lowered volume efficiency. Though this round-wall LPV is lower in volume efficiency that the flat-wall LPV, it has much better volume efficiency than cylinders or multi-lobe tanks.
5.2 Shape flexibility

As mentioned, an LPV may be made with any size and geometric proportions. The reason for this is that the concept is fully modular in the sense that a larger tank simply implies larger size parallel, internal panels and a larger number of such repetitive panels. A very important further property of the LPV concept is that the thickness of the stiffened outer skin does not increase with tank size; it only depends on the internal pressure and spacing between the parallel panels and stiffeners. This is much preferable as compared with cylindrical pressure vessels for which the wall thickness is directly proportional with the tank size as expressed by the radius. For the reason of increasing wall thickness a cylindrical pressure vessels cannot be made very large before they become practically and economically unfeasible; about 1000 m$^3$ seems to be a practical limit. The size of LPVs is not bound by such limitations.
The shape of an LPV is not limited to having essentially a box-like shape. Rounded walls have been mentioned as a variation on this. The internal load carrying structure makes it possible to design largely different tank shapes that fit rather awkwardly shaped rooms within a ship. Figure 7 shows a series of examples of tanks that have been designed and analyzed. Some of these are modifications of the box shape by “cutting off” corners and making wedge forms to adapt to the outer shape of the ship hull. The last example in the figure even shows a wedge shape tank with an “appendix” on the top. Clearly, this is capability goes far beyond what is possible with shell type pressure vessels.

5.3 Examples of ship and offshore applications

A series of different applications have been studied. The first example shown deals with a fuel tank onboard a small vessel as shown in Figure 8. The only practical approach to a small size fuel tank is a Type C pressure vessel since handling of boil-off gas for unpressurized tanks could be too complex and costly. For the available installation space, there is only room for a 4 m³ cylindrical tank whereas the LPV can be fitted to carry 10 m³. The key here is not only that the LPV has a box-like shape, but it has also been given a wedge shape that is adapted to the hull shape. Clearly the LPV solution represents a new opportunity for fueling efficiency and operational range as compared with a conventional cylindrical tank.

Another case studied is to provide 15000 m³ LNG fuel capacity for a large container carrier by use of a single LPV tank. Figure 9 compares this solution to the conventional
solution with 15 cylinders. A major problem with the cylinder solution is that it requires a lot of gross space due to the very low volume efficiency by multiple cylinders whereas the LPV solution has very high volume efficiency and thereby saves a lot of potential cargo space. The difference between the two solutions corresponds to a loss of additional 900 TEU container units by using cylinders which is equivalent to an annual loss of revenue of about 9 million USD per year. Clearly, there are also other advantages by a one tank solution over the multiple cylinder solution; this is caused by less heat ingress and pressure build-up for the one tank solution, less instrumentation and piping, and not at least much simpler operation.

It should also be mentioned that comparisons between using LPV as an alternative to a single Type B (unpressurized) prismatic tank. Both these solutions have good volume efficiency; however, there are main differences in the way the two types of tanks must be operated. For Type B tanks the boil-off gas must be dealt by special equipment and pressurized before feeding the engine. The whole bunkering operation is also quite complicated and prolonged for tanks that can sustain only 0.2 barg pressure. There are also requirements for partial secondary barrier on the insulated fuel tank and thermal insulation and protection of the ship hull. These matters are overcome with type C pressurized tanks which require no boil-off gas compression, no gas process unit and no insulation of the ship hull. In conclusion, the new LPV solution has major advantages over both pressurized cylinders and unpressurized B type tanks.

<table>
<thead>
<tr>
<th>Category</th>
<th>Item</th>
<th>Unit</th>
<th>FSRU With Membrane tanks</th>
<th>FSRU With Bilobe Tanks</th>
<th>FSRU With RW-LPV</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ship</td>
<td>Length O.A.</td>
<td>m</td>
<td>100.0</td>
<td>Ditto</td>
<td>Ditto</td>
</tr>
<tr>
<td></td>
<td>Breadth</td>
<td>m</td>
<td>20.0</td>
<td>Ditto</td>
<td>Ditto</td>
</tr>
<tr>
<td>Cargo tanks</td>
<td>Type</td>
<td></td>
<td>Membrane</td>
<td>Type C</td>
<td>Ditto</td>
</tr>
<tr>
<td></td>
<td>Design pressure</td>
<td>barg</td>
<td>0.7</td>
<td>3.0 (1430%)</td>
<td>Ditto</td>
</tr>
<tr>
<td></td>
<td>Volume of two tanks</td>
<td>m³</td>
<td>30,000</td>
<td>21,800 (71%)</td>
<td>27,300 (92%)</td>
</tr>
<tr>
<td></td>
<td>Weight of two tanks</td>
<td>ton</td>
<td>-</td>
<td>-</td>
<td>1,160</td>
</tr>
<tr>
<td>Other comparison measures</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Inner hull</td>
<td></td>
<td>Required</td>
<td>Not required</td>
<td>Not required</td>
</tr>
<tr>
<td></td>
<td>Secondary barrier</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Gas detection system</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Heated offshore</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Pump tower</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>BOG handling</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Sloshing risk</td>
<td></td>
<td>Probable</td>
<td>No risk</td>
<td>No risk</td>
</tr>
</tbody>
</table>

Figure 10: LPV compared with cylindrical pressure vessels for container ship

Figure 10 shows a final example of alternative tank solutions for a floating LNG storage and regasification unit (FSRU). To the left in the figure is shown a solution where a membrane tank is fitted into the hull of the floater, the middle case is a pressure tank solution with a so-called bilobe tank (two overlapping cylinders), and finally a LPV solution. The membrane tank has the best utilization of the hull space, the bilobe has 27 % less and the LPV
9 % less. The most important features are associated with other aspects such as requirements for secondary barriers, gas leak detection and barriers, pressure pumps and handling of boil-off gas which are matters of disadvantage for the membrane solution. More importantly, FSRU storage tanks will contain all levels of filling under operations. Membrane tanks may have a serious problem with possible damage due to sloshing of the liquid gas; this is a problem that does not exist for the LPVs.

6 SUMMARY

The paper deals with a new type of pressure vessel denoted LPV, Lattice Pressure Vessel. The unique feature with this technology is that it allows for pressure vessels to have a box-like shape, a possibility that previously has not existed for pressure vessels that rather are cylindrical or spherical shell structures. Further, the LPV may even have modified box form with sloping or wedge like shape which means that it can fit and fully utilize an installation space of almost any form. The LPV concept is also fully scalable which means, because of its modular, repetitive internal structure, can be scaled up to almost any size. It can also be dimensioned for almost any desirable pressure. The paper discusses a series of structural, operational and practical advantages by the new LPV technology as compared to conventional technology. This also includes outline of a wide range of interesting and challenging requirements for computational analyses for LNG tanks onboard ships and offshore structures. Finally, the paper outlines and discusses several cases of designs that have been made. The overall conclusion is that the newly available LPV technology represents a major opportunity of improved safety and efficiency over conventional pressure vessel technology; thus, this technology emerges as key for facilitating an infrastructure for more environmentally friendly LNG for power production and for fuels in the transport sector.

ACKNOWLEDGEMENT

The commercial development of the LPV technology from the original patents of KAIST has been jointly conducted by KAIST and POSCO with the major funding from POSCO.

REFERENCES


APPLICATION OF 2D-WAVE SPECTRA TIME SERIES FOR PIPELAYING VESSEL STINGER STRUCTURAL ASSESSMENT

ANDREA DEL GUZZO*, FEDERICO GAGGIOTTI†, CRISTIAN ALBERTO ROSSETTI*, FRANCESCO SAVERIO DI TOMASO† AND ROBERTO BRUSCHI†

*SAIPEM S.p.A. Croatian Branch
Aldo Colonnello 2, 51000 Rijeka, Croatia
e-mail: Andrea.DelGuzzo@saipem.com, CristianAlberto.Rossetti@saipem.com

†SAIPEM S.p.A.
Via Toniolo 1 61032 Fano (PU), Italy
e-mail: Federico.Gaggiotti@saipem.com, FrancescoSaverio.DiTomaso@saipem.com, Roberto.Bruschi@Saipem.com

Key words: Stinger Design, 2D wave spectra

Abstract. In the recent years offshore oil and gas field development activities are moving towards deeper and more remote regions for which high performing installation spread is requested. In those conditions, the offshore pipeline installation in S-lay mode, that is generally preferred being more fast and efficient, presents many challenges in pipeline overbend section and requests a longer curved stinger section to support the pipeline weight during installation. The present paper is focused on stinger structure design and verification and describes the methodology followed to perform advanced global combined hydrodynamic and structural analysis through application of hindcasted 2D wave time series. The analysis is carried out in frequency domain, the vessel motion inducing stinger loads are calculated through application of vessel RAOs and a more realistic description of directional wave energy distribution through 2D sea spectra. Within the proposed methodology a more realistic estimation of dynamic forces vessel motion induced is achieved permitting an higher optimization in material utilization. The practical consequence is that the vessel operational limits can be extended but a more careful management of the offshore operation during execution phase is requested.

1 INTRODUCTION

In the recent years more and more offshore hydrocarbon reservoir have been discovered in challenging areas i.e. deeper locations characterised by harsher environmental condition. The field development includes the installation of infield flow lines and possibly the laying of long or very long export pipelines (sometimes also hundreds kilometres) to deliver the product from offshore to shore. Being faster and more efficient and then less expensive, the S-lay mode is generally preferred but, in those scenarios, presents many challenges in pipeline overbend section and requests a longer curved stinger section to support the pipeline weight during installation [2].

To extend the applicability of S-lay installation mode in deeper areas more performing vessel equipment is requested and particularly longer and lighter stinger ask for a highly
optimized design.

In a challenging offshore market context where the contractor’s investment for new assets is dropping, the request to explore the opportunity to employ the actual capability of the existing installation vessels towards more demanding scenarios, compared to the original design requirements, is also increasing. Currently many vessels equipped for S-Lay mode installation in relatively shallow water are available in the market and the request to investigate the possibility to modify the existing stinger structure making it suitable to the new and more demanding scenarios with minimum investment, as a part of the general system improvement, is becoming more and more frequent. For the above reasons advanced engineering analyses based on more controlled and less conservative approach is mandatory.

Nowadays no international standards provide specific guidelines for the structural design and verification of pipelaying stinger, therefore, robust and extensive engineering studies have to be performed, verified and accepted from the relevant certification bodies. First step for an optimized engineering design is to model realistically the main loads acting on the structure during its service life leaving to the successive structural analysis phase the possibility to reach the requested safety margin through conscious application of partial safety factors on loads and resistance characteristic design values (LRFD method [1]). Focusing on the stinger structure the main loads are coming from the pipeline sustained by rollers, and are primarily dependent on vessel motions [3],[4]. Direct hydrodynamic loads wave and current induced on the stinger in most of the cases can be neglected.

Regarding the simulated vessel behaviour on waves is well known that the real sea conditions generally induces a different vessel motions with respect to the one estimated during the design stage when seastate is theoretically described through synthetic parameter i.e. Significant wave height Hs, peak period Tp and incoming direction [5]. The main reason of this discrepancy must be researched on the classical sea state spectral parametrization including directional spreading formulation. For the range of Hs relevant for the pipelaying operation and considered for stinger design, which are generally significantly lower than survival extreme conditions, the synthetic sea parametrization is not able to fully describe the directional wave energy distribution.

The above is particularly true for the areas such as Offshore Brazil and West Africa where swell and wind sea are contemporarily present and coming from different directions. In those areas the typical representation of total sea brings to incorrect estimation of vessel motions [6].

Nowadays long hindcasting time series that can be considered representative of the waves conditions encountered in the area during the operation are available [7][8] and the more advanced numerical model can provide also a detailed directional distribution of the wave energy i.e. 2D spectra [9]. All these information can be utilized as input to simulate more realistically the vessel motions and the corresponding stinger induced loads.

As an example Figure 1 presents for a sea state of Hs=1.5m a comparison between theoretical and actual spectrum. The directional distribution of the wave energy is quite different for the two cases and it is reflected in vessel motion estimation. Referring to a mono-hull pipelaying vessel, theoretical sea spectra induces higher vessel motions if compared with those corresponding to actual sea state. The vessel motions can be calculated in frequency domain through Response Amplitude Operators (RAOs) and the statistical maxima can be estimated and applied as input for the structural verification of the stinger structure. It is worth to underline that the present
methodology for vessel motion estimation is general, and can be applied every time more rigorous vessel motion evaluation is requested. For the structural verification keeping as input the vessel motions calculated as above, it is still allowed to follow any recognised international standard or code check criteria without any particular constraint.

With proposed methodology a more consistent and effective factor of safety application is achieved resulting in safer and more optimized material utilization. In particular for the design of new stingers the final structure is safer, slender and lighter with positive impact on operational performances and on material and fabrication costs.

For stinger already in operation the new approach gives the opportunity to better assess the real capability of structure designed with different philosophy with possibility on extending its service life pushing toward more demanding scenarios e.g. deeper water of harsher areas.

The present paper is organized as follows. Section 2 provides the theoretical background on which the calculation is based. Section 3 describes how the general theory can be applied for the stinger structural analysis. Section 4 contains a typical example showing how the process can be a valuable way to perform global verification of an existing stinger structure confirming the possibility to extend the pipelaying water depth range towards deeper areas.

Discussion on the main outcomes and conclusions are collected in the last chapter.

2 THEORETICAL BACKGROUND

The structural verification of any offshore structure rigidly connected on a floating vessel e.g. FPSO topside or jacket transported on cargo barge is typically performed calculating
statistical maximum of the vessel motions (displacement and acceleration) induced by 3 hours extreme design sea state. The extreme acceleration can be calculated through vessel Response Amplitude Operators (RAOs) that describe for each of the 6 Degree of Freedom (DOF- 3 translations and 3 rotations), for each frequency and for each vessel-wave relative direction how the vessel moves if excited by 1m amplitude regular wave.

The RAOs are generally provided in to the vessel Centre of Gravity (CoG) but can be transferred to any point under the hypothesis of rigid body motion. For a point with coordinates a, b, c with respect to CoG the translation is defined by:

\[
\begin{align*}
X_{abc} &= X_{CoG} + c_0 - b_0 \\
Y_{abc} &= Y_{CoG} - a_0 \cos \theta_y + a_0 \sin \theta_y \\
Z_{abc} &= Z_{CoG} + b_0 \cos \theta_z - a_0 \sin \theta_z 
\end{align*}
\]  

where \( \theta_x, \theta_y, \theta_z \) are the rotation angles around the three coordinates axes. Therefore, given the motion RAOs at the CoG defined as:

\[
X_{CoG} = X(\omega, \theta) \cdot e^{i(\omega t + \phi)}
\]

the motion, velocity and acceleration RAOs for each of the 6 DOF at any generic point can be calculated as:

\[
\begin{align*}
X_{abc} &= X_{CoG}(\omega, \theta) \cdot e^{i(\omega t + \phi)} \\
\dot{X}_{abc} &= i\omega X_{abc}(\omega, \theta) \cdot e^{i(\omega t + \phi)} \\
\ddot{X}_{abc} &= -\omega^2 X_{abc}(\omega, \theta) \cdot e^{i(\omega t + \phi)}
\end{align*}
\]  

The acting force in vector form \( \vec{F}_1 \) on i-th beam-like body positioned at coordinates a,b,c with respect to CoG can be defined:

\[
\vec{F}_{Dyn,i} = m_i \ddot{X}_{abc} + \rho V C_a (\dot{X}_{abc} - \dot{U}_{abc}) + \frac{1}{2} \rho C_d A (\ddot{X}_{abc} - \ddot{U}_{abc}) \cdot \ddot{U}_{abc}
\]

Where:

\( \vec{F}_{Dyn,i} \) = force applied to the beam-like element (normal to the beam axis); 
\( m_i \) = structural mass; 
\( \rho \) = water density 
\( V \) = displaced volume 
\( A_i \) = cross area of the beam-like body; 
\( C_a \) = added mass coefficient; 
\( C_d \) = drag coefficient; 
\( \dot{X}_{abc} \) = velocity of the beam-like body (normal to the beam axis); 
\( \ddot{X}_{abc} \) = acceleration of a beam-like object (normal to the beam axis); 
\( \dot{U}_{abc} \) = velocity of the water particle (normal to the beam axis); 
\( \ddot{U}_{abc} \) = acceleration of a beam-like object (normal to the beam axis);
\( U_{abc} \) = acceleration of water particle (normal to the beam axis).

The loads calculated as per the above formulation are applied, and the equilibrium and compatibility equations are solved for each frequency. The transfer functions of the force (FAO) for each i-th beam element are then obtained as shown in figure 2.

**Figure 2:** Force Amplitude operator (FAO) – Axial - for direction 0-360deg.

Figure 2 presents the FAO for the axial forces acting on a structural member along its axis. The FAO and the sea state spectra can then be combined to calculate the force spectrum.

**Figure 3:** Axial force spectrum in a structural member as a function of a directional wave spectrum

As an example Figure 3 shows how can be calculated the axial dynamic load \( F(\omega, \theta) \) induced by vessel motion on a structure specific location. It represents a graphical schematization of the Eq. (7) and (8) for a specific wave spectrum.
The statistical expected most probable maximum dynamic force is calculated for N hours sea state following Rayleigh statistic formulation:

\[
F_{\text{extr,mean}} = F_{\text{sig}} \frac{1}{\sqrt{2 \pi}} \left( \frac{\ln \left( \frac{N \cdot 3600}{T_2} \right)}{\ln \left( \frac{N \cdot 3600}{T_2} \right)} + \frac{0.2886}{\ln \left( \frac{N \cdot 3600}{T_2} \right)} \right) \cdot CF
\]  

(10)

\[
F_{\text{extr,mode}} = F_{\text{sig}} \frac{1}{\sqrt{2 \pi}} \left( \frac{\ln \left( \frac{N \cdot 3600}{T_2} \right)}{\ln \left( \frac{N \cdot 3600}{T_2} \right)} \right) \cdot CF
\]  

(11)

where the zero up-crossing period \( T_2 \), the bandwidth correction factor \( CF \), the \( \varepsilon^2 \) spectrum broadness parameter and \( m_n \) the even moments of the spectrum, are:

\[
T_2 = \frac{m_2}{m_0}
\]  

(12)

\[
CF = 1 - \varepsilon^2
\]  

(13)

\[
\varepsilon^2 = \frac{m_4}{m_2^2}
\]  

(14)

\[
m_n = \int_0^{2\pi} S_f(\omega, \theta) d\omega d\theta
\]  

(15)

As per Eq. (8), the total significant force is calculated as the result of a double integration on the frequencies and directions. The final force acting on the stinger is calculated adding the static (load for zero environments) and dynamic component:

\[
F_{\text{Max}} = F_{\text{extr,mode}} + F_{\text{Stat}}
\]  

(16)

3 CASE STUDY: VERIFICATION OF EXISTING STINGER STRUCTURE

The above methodology has been applied to verify the suitability of an existing stinger structure, initially designed for S-lay operation in relatively shallow water areas, on laying in deeper areas with harsher environment. The laying scenario refers to 14” pipe installation in 500m water depth.

The stinger is composed by 2 rigid ramps (from now on referred as truss and intermediate), and a floating section connected to intermediate. Stinger ramps have 3 rollers on the truss section, 1 roller on the intermediate section and 4 rollers on floating ramp. All the stinger sections are reticular structures and connected to a laying barge 120m length and 33m wide for which the main characteristics and operative laying loading conditions are listed in Table 1.
### Table 1: Laying barge main characteristics

<table>
<thead>
<tr>
<th>Floating Position</th>
<th>Displacement</th>
<th>21815.0 [tonne]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Draft At AP</td>
<td>6.52 [m]</td>
<td></td>
</tr>
<tr>
<td>Draft At Midship</td>
<td>5.94 [m]</td>
<td></td>
</tr>
<tr>
<td>Draft At FP</td>
<td>5.36 [m]</td>
<td></td>
</tr>
<tr>
<td>KMT</td>
<td>16.28 [m]</td>
<td></td>
</tr>
<tr>
<td>KG</td>
<td>9.44 [m]</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Natural Periods</th>
<th>Heave, Tn,3</th>
<th>8.4 [s]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Roll, Tn,4</td>
<td>11.7 [s]</td>
</tr>
<tr>
<td></td>
<td>Pitch, Tn,5</td>
<td>8.0 [s]</td>
</tr>
</tbody>
</table>

#### 3.1 Numerical model

The dynamic load induced by the pipeline during laying operation has been applied as static forces acting on the roller locations. On the fixed ramps the maximum expected dynamic forces on rollers are applied assuming that the maximum loads on the stinger due to pipeline occur at the same instant. On the floating ramp, to properly reproduce the behavior during operation, only pipeline’s static loads have been considered.

The mass and buoyancy are correctly reproduced through combined structural and hydrodynamic model. For this latter the hydrodynamic forces on slender element has been estimated applying Morison theory and for large volume element, like barge, through potential panel method. The stinger is connected to the laying barge through:

- Lower connection: hinge allowing only rotation on the vertical plane;
- Upper connection: boom braces hinged both side to barge and stinger allowing only rotation around hinge axes.

Boundary conditions applied in the model are chosen to properly reproduce the transfer of forces between elements. Pinned connections are applied on the upper and lower hinges on the vessel and for the connection between the intermediate and the floating ramp. For the numerical calculation purposes only, in particular to make the structure properly constrained, an extra support is requested to the floating ramp tip. For each analysis the ballast water in the floating ramp has been defined to assure in each laying scenario a negligible reaction on the tip support. Figure 4 presents the combined structural/hydrodynamic model of the stinger and the boundary conditions applied.

For the hydrodynamic model the stinger structure has been modeled through tubular sections and rollers supporting pipelines are included, so that correct weight and drag is modeled. For both elements, tubular section and rollers, the applied drag and added mass coefficient are $C_D = 0.8$ and $C_A = 1.0$ respectively.
The analysis has been performed through SESAM (DNV) suite [10], in particular GeniE, WADAM, SESTRA and Xtract packages have been used for design of the structure, Hydrodynamic loads calculation, for structural analysis and results post-processing, respectively.

The combined analysis with barge and stinger model has been performed to calculate the FAO. For the specific case, only FAOs from 0 to 360 deg with a step of 22.5 deg and 0.5s of resolution for the jacking booms have been calculated. A dedicated analysis has been performed to clearly identify the structural element that in a hierarchical scale for all the loading conditions is the first reaching the structural limit. In this way a clear and well defined criterion has been identified and corresponds to the maximum structural capacity of jacking boom elements.

All the 2D wave spectra, 50 years long time series of 3 hours sea state, for the representative laying scenario has been applied. The forces induced by vessel motions on the booms are computed considering the relative angle between vessel and incoming wave direction assuming, for each section of the pipe to be laid, the realistic vessel heading.

It is worth to note that for the case where a unique screening criterion cannot be identified the described methodology is still applicable but the calculation has to be repeated for all possible limiting conditions e.g. structural integrity for various structural elements or nodes.

The calculation effort will increase and final acceptable sea states are those that contemporarily satisfy all possible limiting criteria.

3.2 Case study results

Making reference to the model described above for all the events in the time series the forces on jacking booms have been calculated. The loads from pipeline statically applied to the stinger in rollers location for the analyzed scenario are given in Table 2. The limiting force for the jacking booms is assumed to be 400t each. In Figure 5, the stinger starboard and portside jacking booms FAOs for the analyzed laying scenario are shown.
Table 2: Pipe loads on stinger system rollers-rollers numbering from vessel barge hinges.

<table>
<thead>
<tr>
<th>CASE</th>
<th>CRITERIA CHECK</th>
<th>Stinger</th>
<th>Intermediate</th>
<th>Floating</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>TR1</td>
<td>TR2</td>
<td>L1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>[m]</td>
<td>[m]</td>
<td>[m]</td>
</tr>
<tr>
<td>Normal lay (scenario 50m-0°)</td>
<td>Static</td>
<td>16.5</td>
<td>10.5</td>
<td>18.5</td>
</tr>
<tr>
<td></td>
<td>Dynamic</td>
<td>41.0</td>
<td>14.7</td>
<td>49.6</td>
</tr>
<tr>
<td></td>
<td>Total</td>
<td>77.5</td>
<td>25.2</td>
<td>88.1</td>
</tr>
</tbody>
</table>

Figure 5: Force Amplitude Operator of axial force for Starboard and Portside jacking boom

Applying the FAO and all the 2D spectra sea state in the 50 year long time series the axial forces on the booms are calculated. The resulting axial loads are plotted against the Hs, Tp, spreading and relative direction (Figure 6).
Referring to the installation scenario of laying 14" pipeline in 500m water depth to evaluate the improvement in terms of installation performances, the same stinger structural assessment has been repeated considering the sea states time series provided in term of synthetic parameter Hs, Tp and regular wave approach (H_{max}=H_{reg}=Hs*1.83). In the standard approaches a long-crested sea or regular wave no directional distribution of the sea state energy, is accounted for. Comparing the results of the proposed methodology with the standard ones i.e. regular and irregular wave approaches the improvement in terms of stinger performance can be evaluated.

Figure 7 presents the axial loads on booms calculated for the considered methodologies.

Referring to the Hs for the considered laying scenario, assuming acceptable an operative limit not smaller than 65% is observed that a sea state of Hs of 1.46m can be assumed as upper limit. For the same Hs the classical approach, based on synthetic parameter Hs/Tp, gives around 5% operability, too low to be considered acceptable. Same if regular wave approach would be applied. With the improved methodology 60% of actual operative cases that with the classical approaches resulted not operative are then included. In other words the overall operability increase achieved with the applied methodology is around 60%.

Regarding the allowable sea states Hs \leq 1.0m result operative cases independently on the verification methodology applied. Hs \geq 1.2m and Hs \geq 1.1m are never acceptable if irregular or regular wave approach is respectively considered. From the analysis with 2D spectra comes
that for Hs up to 1.46m the stinger structure is not a limiting factor.

For the specific laying scenario the limit of the installation spread is Hs ranging between 1.4m and 1.5m. So Hs=1.45m can be reasonably assumed as the overall limit for the laying operation.

Focusing the attention only to the stinger structure, accepting a reduced operability, also higher limits can be considered acceptable but in this case under certain conditions.

In pipelaying installation it can be acceptable for short weather windows operations such as initiation or final laydown but not for normal laying during long laying campaign for which low operability corresponds to unacceptable cost and operational risk increase due to long waiting on weather and increase of number of pipe abandonment and recovery operations.

Considering acceptable an operative level of 40% the new approach allows including sea states with Hs up to 1.55m. For the same Hs the regular and irregular wave approaches gives 16% and 26% lower Hs limits.

Within the 2D spectra approach the limits are increased since also during the engineering study phase the dependence of vessel motion on wave height, peak period, and directional energy distribution can be properly account. The same is not possible in case classical regular and irregular wave based methodologies are applied.

From the above a more careful management during execution phase is requested and specific operational procedures, reliable and high quality weather service forecast during installation execution are needed.

4 CONCLUSIONS

The present paper presents and describes new methodology to be applied to verify the stinger structure capability accounting for a more realistic description of sea state. The methodology is general and can provide an appreciable benefit for all the cases when floating structures and floating structure’s equipment design loads are mainly related to dynamic motions.

The method is based on the proven assumption that the classical way to describe the sea state through synthetic parameter Hs and Tp, inducing vessel motions, in most of cases barely reproduce the realistic sea condition. This is particularly true for area such as West Africa or Brazil affected by crossed sea state i.e. Wind Sea and Swell contemporary present. The consequence is that extreme vessel motions considered for equipment designed and calculated during design phase are overestimated with respect to actual on board registration.

Nowadays state of art hindcasting numerical models are able to provide a detailed and more realistic description of the sea state including a more realistic directional distribution of wave energy. The frequency domain analysis based on the 2D sea spectra time series allows a better estimation of the main loads for the vessel motion.

The proposed methodology bring an improvement of the design load estimation without any variation on the next structural design phase. The design approach and corresponding verification criteria as per most appropriate international standard can still be followed.

For the present paper the general methodology accounting for hindcasting sea state 2D spectra has been applied for the structural verification of stinger already in operation. The aim was to demonstrate that the structure can be suitable for the installation on more demanding scenarios comparing to those initially accounted for the original design.

A practical example has been provided showing a significant improvement of around 60%
in term of pipe laying operability. The methodology can be followed to optimize the design of new build stinger allowing more slender, light and definitively more performing structure.

The real benefits are in material and fabrication costs with additional saving on management and system maintenance during service life.

REFERENCES

STATIC ANALYSIS OF A COMPOSITE WIND TURBINE BLADE USING FINITE ELEMENT MODEL

Meltem ÖZYILDIZ¹, Demirkan COKER²

¹ Graduate Student, Dept. of Aerospace Engineering Department, METU, 06800 Çankaya Ankara/TURKEY, e-mail: mozyildiz@aselsan.com.tr
² Assoc. Prof. Dr, Dept. of Aerospace Engineering Department, METU, 06800 Çankaya Ankara/TURKEY, e-mail: coker@metu.edu.tr

Keywords: Composite Materials, Finite Element Analysis, Static Analysis, Wind Turbine Blade

Abstract: This study is presented here that the stress characteristics of an existing 5-meter composite wind turbine blade for 30 kW wind turbine designed for METUWIND is known by using finite element method. Modal and static analysis is performed in order to obtain static and dynamic behavior of the blade. To perform analysis, the geometric three-dimensional model of the blade is obtained by using two-dimensional drawings of the blade. After geometric modeling of the blade, the materials that are used in blade structure are applied to Ansys ACP. Then, the blade structure model is adapted a finite element solver, Ansys Workbench. Finally, loading conditions are applied along the blade and the results are obtained. The results of this study indicate that the internal flange is the main force-supporting part, while other parts of the blade are mainly keeping the blade stable.

1. INTRODUCTION

The long-term reliability of wind turbines is very important for sustainable and economically viable wind energy utilization. Therefore, wind turbine designs must be optimized to minimize costs and maximize lifetime. To do these requires performance and durability characteristics of wind turbine materials, components and structures must be understood extensively. The most critical component of a wind turbine is the composite rotor blade. A rotor blade failure can have a significant impact on turbine downtime and safety. To avoid a blade failure, knowing the strength of the rotor blades is essential. Hence, validation of blade resistance must be checked by structural testing and/or analysis [1]. For instance, a full-scale 34-meter composite wind turbine blade was tested to failure under flap-wise loading and simulated in finite element calculations [2]. Furthermore, to evaluate a proposed medium scale composite wind turbine blade, structural analysis was performed by using the finite element method [3]. However, the
structural testing methods such as full-scale testing of the blade are expensive and troublesome due to constructing a test set-up. The structural analysis of a blade is necessary before testing of the final design. Structural analysis is usually conducted to determine stress distribution, deflections, modal analysis and fatigue behavior. Additionally, to investigate natural frequency of a wind turbine blade it is essential to consider the dynamic characteristics of the turbine like rotational speed, wind speed [4]. The most acceptable and common analysis method for the structural validation of a wind turbine blade is finite element modeling.

The objective of this paper is to develop a finite element model to analyze an existing 5-meter wind turbine blade to check the strength of the blade subjected to static and dynamic loading. The existing blade was designed as part of a project between METUWIND – Center of the Wind Energy and Core Team of the University of PATRAS according to IEC 61400-2 [5]. The blade was designed for a wind turbine that has 30 kW nominal power capacity at 10 m/s wind speed. According to the wind turbine characteristics, the blade optimized aerodynamic and geometric design was finalized by blade manufacturer.

The existing blade consists of five main parts: Suction Side, Pressure Side, internal flange, “hat shaped” chassis, and flange; see Figure 1. Total length of the blade is 5 meters. Two shear webs part of a “hat shaped” chassis and the internal flange are placed in the blade, starting at 0.5 m away from the root and extending up to 4.0 m.

2. FINITE ELEMENT MODEL

2.1. Geometric Modeling

The two-dimensional blade drawings which include the blade aerodynamic design details such as cord length and twist angle along the blade were provided by the blade manufacturer. By
using these given two-dimensional blade drawings, the three-dimensional geometric modeling of the blade is performed in NX 10.0 environment [7]. The geometric modeling of the blade and some blade sections are given in Figure 2.

![Figure 2. Geometric modeling of the blade.](image)

### 2.2. Material Modeling

Materials in blade structure are composed of gel coat, steel and composite laminates. In the existing blade, gel coat, chopped strand mat (CSM 300), steel, polymetric foam (Divinycell H45) and two fiberglass composite materials are used. These glass/epoxy composites are: unidirectional laminate and tri-axial laminate. The unidirectional laminate is called METYX L600E10C-0 of 623 g/m2 with parallel continuous fibers. The second glass fabric, METYX XL800E10F-[0/45/-45] of 835 g/m2, is a tri-axial architecture with fibers in the 0, +45° and -45° directions in a ratio of 2:1:1. These layers have the same properties but different thicknesses.

The laminate materials of the blade are named as follows:

i. Unidirectional laminate with a thickness of 0.716 mm
ii. Tri-axial laminate [0/45/-45] with a thickness of 
   h0TRI=0.483 mm, 
   h+45TRI=0.238 mm and h-45TRI =0.238 mm.
iii. Steel with a thickness of 5.3 mm
iv. Gel Coat with a thickness of 0.9 mm
v. CSM 300 with a thickness of 0.358 mm
vi. The polymeric foam, Divinycell H45 of DiAB group, with thickness varying from 5 mm to 10 mm.

Table 1 lists the experimental material properties of the blade materials [8].
Table 1: Experimental mechanical properties of the UD ply

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Unidirectional Laminate</th>
<th>Steel</th>
<th>Gel Coat</th>
<th>CSM 300</th>
<th>Divinycell H45</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density, $\rho$</td>
<td>[kg/mm$^3$]</td>
<td>1896</td>
<td>7850</td>
<td>1200</td>
<td>1896</td>
</tr>
<tr>
<td>Thickness, $h$</td>
<td>[mm]</td>
<td>0.716</td>
<td>5.3</td>
<td>0.9</td>
<td>0.358</td>
</tr>
<tr>
<td>$E_1$</td>
<td>[GPa]</td>
<td>12.17</td>
<td>210</td>
<td>1.95</td>
<td>4.47</td>
</tr>
<tr>
<td>$E_2$</td>
<td>[GPa]</td>
<td>4.47</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\nu_{12}$</td>
<td>[GPa]</td>
<td>0.14</td>
<td>0.3</td>
<td>0.17</td>
<td>0.14</td>
</tr>
<tr>
<td>$G_{12}$</td>
<td>[GPa]</td>
<td>1.38</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$X_T$</td>
<td>[MPa]</td>
<td>191.73</td>
<td>581.8</td>
<td>35.29</td>
<td>16.86</td>
</tr>
<tr>
<td>$X_C$</td>
<td>[MPa]</td>
<td>101.16</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$Y_T$</td>
<td>[MPa]</td>
<td>16.86</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$Y_C$</td>
<td>[MPa]</td>
<td>50.41</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$S$</td>
<td>[MPa]</td>
<td>11.29</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The lamination plan of the pressure and suction sides of the blade can be described below:

- The outer surface of the blade is covered with transparent Gel Coat and a layer of chopped strand mat, 300 g/m$^2$ CSM 300.
- The root part of the blade is composed of unidirectional laminate, tri-axial laminates and steel.
- The lay-up sequence for the pressure and suction side differs only in the area from 1.25 m to 2.0 m where an additional unidirectional glass fabric was placed in the suction side of the blade.
- The Divinycell H45 foam used in the trailing edge is of 10 mm thickness in the area from 0.7 m to 2.0 m and 5 mm thickness from 2.0 m to 3.0 m.

The composite layups of the blade were defined in ANSYS Composite PrepPost (ACP) [9]. Due to different lamination plan for different blade section, the thickness of the blade varies along the blade. The composite layer thickness changes can be clearly seen from Figure 3. The dark blue areas are the thinnest regions while light blue areas (“hat shaped” chassis area) are thicker regions. The thickest region is the orange region (area from 0 m to 0.2 m) as expected.

Figure 3. The composite layer thickness for different blade section.
After material modeling of the blade was finished in Ansys ACP, the blade was meshed entirely with 20013 layered shell elements and 42975 nodes in ANSYS Workbench [10]. The fine mesh density was chosen on the blade training edges because of investigating more detailed stress distribution on these areas.

![Figure 4. (a) The mesh all around the blade (b) The mesh around the tip of the blade.](image)

3. RESULTS

The result for the modal analysis is presented in Section 3.1 and static analysis is in Section 3.2.

3.1. Modal Analysis

Modal analysis proves to determine dynamic characteristics of wind turbine blades. To estimate the mode shapes and natural frequencies of the existing blade, a modal analysis was performed in ANSYS Workbench. In the modal analysis, the blade was fixed at the root by fixing all degree of freedom. Table 2 presents the results of the modal analysis that calculated the natural frequencies for bending modes in the edgewise and flapwise directions. The mode shapes of the blade in both directions can be seen in Figure 5.

<table>
<thead>
<tr>
<th>Mode Shapes</th>
<th>Natural Frequencies [Hz]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1st Mode Shape</td>
<td>1st Flapwise Bending</td>
</tr>
<tr>
<td>2nd Mode Shape</td>
<td>2nd Flapwise Bending</td>
</tr>
<tr>
<td>3rd Mode Shape</td>
<td>3rd Flapwise Bending</td>
</tr>
<tr>
<td>4th Mode Shape</td>
<td>4th Flapwise Bending &amp; 1st Edgewise Bending Coupling</td>
</tr>
<tr>
<td>5th Mode Shape</td>
<td>5th Flapwise Bending &amp; 2nd Edgewise Bending Coupling</td>
</tr>
<tr>
<td>6th Mode Shape</td>
<td>3rd Flapwise Bending &amp; 1st Torsion Coupling</td>
</tr>
</tbody>
</table>

![Table 2: Results of modal analysis.](table)
Figure 5. Mode shape of the blade.
3.2. Static Analysis

To investigate the blade behavior under static loading, loading is applied to the blade in both edgewise and flapwise directions. In these directions, loads were selected according to blade design conditions. These design loads were obtained from the worst case scenario of the wind turbine operating conditions. The extreme force distributions on the blade are presented in Figure 6. These forces were placed at 21 stations along the blade. The edgewise and flapwise analysis results are presented in the following sections.

![Force distribution over blade radial position](image)

**Figure 6. Force distribution over blade radial position**

3.2.1. Edgewise Analysis

Blade mass and gravity causes bending moment in the edgewise direction of the blade. In the edgewise analysis, the blade was fixed at the root by fixing all degree of freedom. The weight due to standard earth gravity was added to the extreme design loads given in Figure 6 at the stations along the blade. The deformed geometry and the stress distribution are given in Figure 7 and Figure 8, respectively.

The total deformation occurs at the tip of the blade as expected and has a value of 83 mm in the edgewise direction.
The maximum equivalent stress is approximately 77 MPa at 4.0 m from the root. The detailed stress distribution in the blade cross-section at 4.0 m from the root is given in Figure 9. The stress in the pressure and suction sides reaches the maximum value of 26 MPa at the leading edge. In the chassis, the stress is the maximum at the corner of the shear webs and the spar cap near the leading edge. Finally, absolute maximum stress of 77 MPa occurs in the internal flange at the suction side.
3.2.2. Flapwise Analysis

The most significant bending moments in the blade occur in the flapwise direction. Therefore, the blade must have high rigidity in that direction. To evaluate the rigidity, the blade was constructed at the root end surface by fixing all six degree of freedom and flapwise loads are given in Figure 6 were applied along the blade. The deformed geometry and the stress distribution are given Figure 10 and Figure 11, respectively.

![Figure 10. Total deformation under loading in flapwise direction.](image)

The total deformation is almost 1.0 m at the tip of the blade.

![Figure 11. Stress distribution under loading in the flapwise direction in the suction and pressure sides.](image)

The maximum equivalent stress is approximately 530 MPa at 4.0 m from the root. The detailed stress distribution in the blade cross-section at 4.0 m from the root is given in Figure 12. The stress in the pressure and suction sides reaches the maximum value of 78 MPa at the leading edge. In the chassis, the stress is the maximum at the corner of the shear webs and the spar cap near the leading edge. Finally, absolute maximum stress of 530 MPa occurs in the internal flange at the suction side.
4. CONCLUSIONS

Within the scope of this article, an existing 5-meter long wind turbine blade was modeled and analyzed with finite element program using Ansys Workbench environment. Static and modal analyzes were performed to investigate edgewise and flapwise loading behaviors. The loads were selected according to blade design conditions.

The lowest frequency from the modal analysis was found to be 6.1 Hz for first mode. The primary modes are found to be dominantly of either in the flapwise or edgewise direction. The edgewise and flapwise coordinates are coupled in the mode shapes, and the coupling is not negligible.

Both edgewise and flapwise static analysis shows that the whole stress level at the internal flange is higher than that of chassis, pressure and suction sides of the blade. It indicates that the internal flange is the main force-supporting part, while other parts of the blade are mainly keeping the blade stable.
In the light of the points having been put forward so far, it can be concluded that main function of the chassis is not to support bending loads, but to resistance shear deformation, and to enhance the structural stability of the blade.

5. REFERENCES


INDEX AUTHORS
This volume contains the full papers of the VII International Conference on Computational Methods in Marine Engineering (MARINE 2017) held in Nantes, France on May 15-17, 2017.