11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
E. Oñate, J. Oliver and A. Huerta (Eds)

WAKE SIMULATION OF A MARINE PROPELLER

E. GUILMINEAU, G.B. DENG, A. LEROYER, P. QUEUTEY, M. VISONNEAU AND J. WACKERS

Laboratoire de recherche en Hydrodynamique, Energétique et
Environnement Atmosphérique (LHEEA), CNRS 6598
Ecole Centrale de Nantes
1 rue de la Noë, B.P. 92101, 44321 Nantes Cedex 3, France
e-mail: Emmanuel.Guilmineau@ec-nantes.fr

Key words: Propeller, RANS, DES, Turbulence closures, Tip vortex.

Abstract. The flow past a rotating marine propeller is analyzed with the aim of establishing capabilities of different turbulence modelling approaches. Two RANSE models are used: the k-ω SST of Menter and an anisotropic two-equation Explicit Algebraic Reynolds Stress Model (EARSM). A DES approach based on the k-ω model is also used. The numerical simulations with the RANSE models give a good prediction of the thrust and torque. However, the wake of the propeller is too dissipate and the instabilities of the wake are not predicted. On the contrary, DES approach can allow to capture the evolution of the tip vortices and predicts the onset of instabilities in the wake.

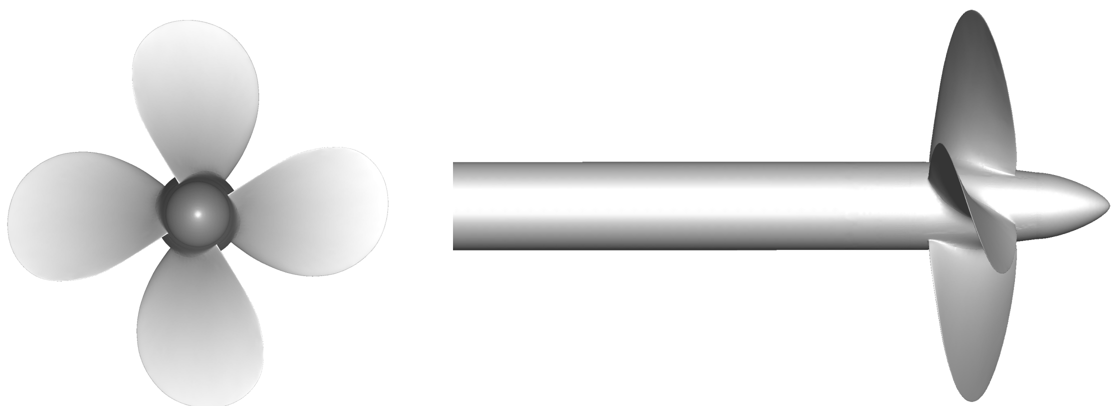
1 INTRODUCTION

The prediction of the fluid dynamic interaction between propellers and the hull is very important for the improvement of ship performance since this interaction is directly related to vibrations, noise and propulsion performances. In this context, the demand for the improvement of performances implies a rising interest in the development and application of detailed numerical tools. The physical mechanisms that characterize the interaction between the propeller and the hull are very complex. However, even in the simpler case of an isolated propeller in a uniform flow, called open water conditions, one is confronted with severe numerical and physical challenges. A comprehensive description of the state of the art can be found in Felli et al. [1], who experimentally investigated the flow around a propeller in a water channel. These authors studied the mechanisms that trigger the instability of wake and investigated the depedence of the vortex pairing and grouping on the mutual vortex distance.

In this paper, the flow past a propeller model is analyzed with the aim of establishing limits and capabilities of different turbulence modeling approaches. These models are used with the ISIS-CFD unstructured finite-volume solver.

2 TEST CASE

The propeller geometry is the INSEAN E779A model, i.e. a four blade, fixed-pitch, right-handed propeller characterized by a nominally constant pitch distribution and a very low skew, see Figure 1.



(a) Front view (b) Side view

Figure 1: Views of the propeller model

In this paper, the rotational speed of the propeller is kept fixed to a value of $n = 25$ rps and the different advance coefficients $J = U_\infty/nD$ are obtained by changing the inflow velocity U_∞. The Reynolds number $\text{Re} = 1.78\times10^6$ is based on the radius of the propeller ($L_{ref} = R = 0.1135$ m) and the velocity of the tips of the blades ($U_{ref} = n\pi D \simeq 17.829$ m/s).

3 FLOW SOLVER

ISIS-CFD, developed by the Ecole Centrale de Nantes and CNRS and available as a part of the FINE/Marine computing suite, is an incompressible unsteady Reynolds-averaged Navier-Stokes (URANS) method. The solver is based on the finite volume method to build the spatial discretization of the transport equations. The unstructured discretization is face-based, which means that cells with an arbitrary number of arbitrarily shaped faces are accepted. A second order backward difference scheme is used to discretize time which is treated in a fully implicit manner. Thus, there is no stability limit on the CFL number. The solver can simulate both steady and unsteady flows. The velocity field is obtained from the momentum conservation equations and the pressure field is extracted from the mass equation constraint, or continuity equation, transformed into a pressure equation. In the case of turbulent flows, transport equation for the variables in the turbulence model are added to the discretization. A detailed description of the solver is given by Duvigneau et al. [2] and Queutey and Visonneau [3].

The method features sophisticated turbulence models: apart from the classical two-equation k-ε and k-ω models, the anisotropic two-equation Explicit Algebraic Reynolds Stress Model (EARSM), as well as Reynolds Stress Transport Models, are available, see Duvigneau et al. [2] and Deng and Visonneau [4]. All these are RANS models. Recently, a Detached Eddy Simulation (DES) approach has been introduced, see Guilmineau et al. [5].

4 NUMERICAL SIMULATION SET-UP

The computational domain consists of a cylindrical domain, whose diameter is 3 times the propeller diameter, and length is 9.2 times the propeller diameter. A constant free-stream velocity boundary condition is specified at the inlet and lateral boundaries. At the outlet, the pressure is imposed.

The computational mesh is created with the HEXPRESS software, an automatic unstructured mesh generator. This software generates meshes containing only hexahedrons. The mesh consists of 21.4×10^6 cells. The number of faces for each blade is approximately 38 100. The average wall normal resolution on the blades is $y^+ = 0.6$ with a maximum, around the tip, in the order of 1.8.

5 RESULTS

5.1 Thrust and torque

In order to compare different approaches for the prediction of the flow around the propeller, the open water characteristics of the model are investigarted. Several numerical results for different values of the advance coefficient J are compared with the experimental data, see figure 2. Concerning the thrust Kt and the torque Kq, the predictions obtained with the different turbulence models differ by less than 5% for the low values of the advance coefficient and by less than 3% for the high value of J. From the point of view of global quantities estimation, it therefore appears that the use of a more accurate turbulence model is not justified.

5.2 Flow field in the wake

However, it is crucial to evaluate the ability of the computational model to create and convect the vortical structures which are associated with a propeller flow. The analysis of the flow field is carried out for two values of the advance coefficient, namely $J = 0.71$ and 0.45 which are considered in order to try to reproduce some of the experimental findings of Felli et al. [1]. At $J = 0.71$, the grouping process of the tip vortex filament occurs at $x = 12.65R$, which is far in the wake. In order to capture the first stages of the vortex coupling phenomenon described in the experimental work, a second value of the advance coefficient is chosen, $J = 0.45$.

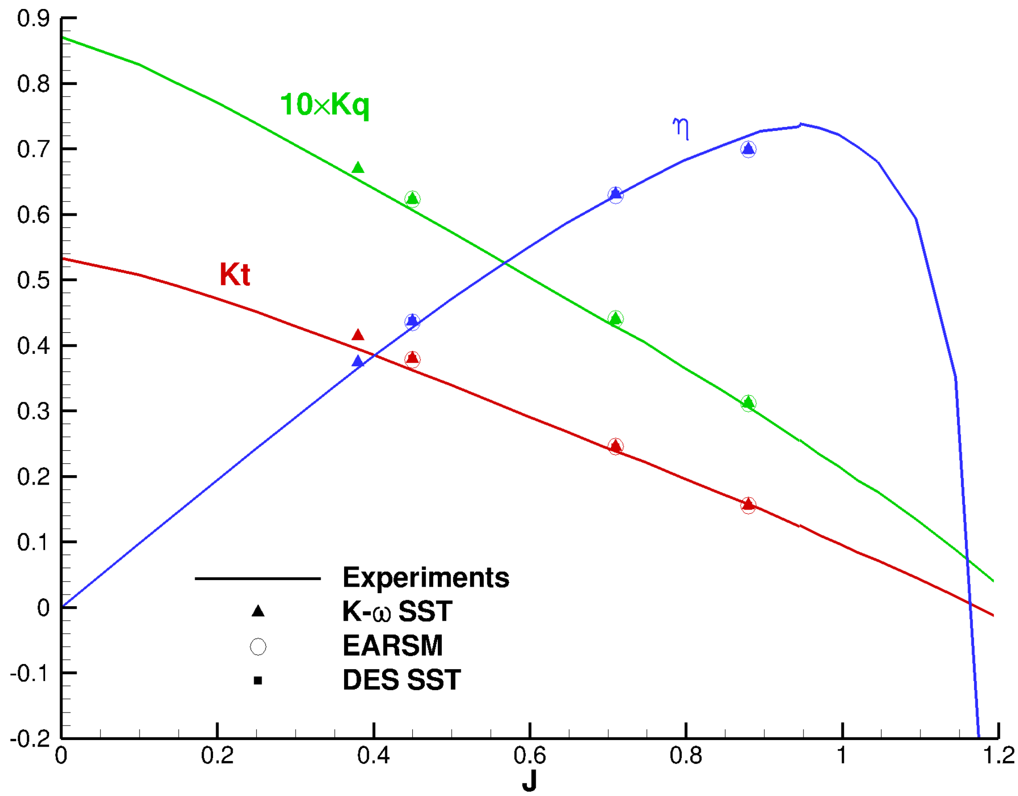


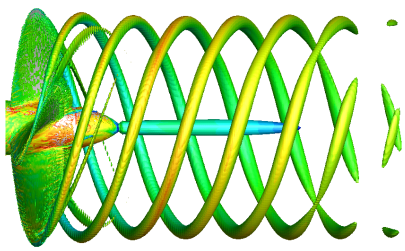
Figure 2: Open water characteristics of the E779A propeller

5.2.1 J = 0.71

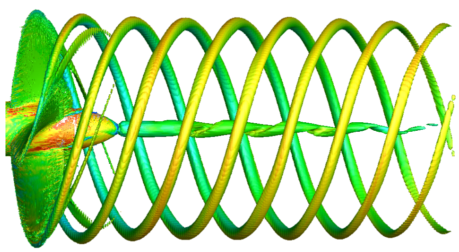
A general overview of the wake of the propeller is given in figure 3 which presents the surface of the non-dimensional value λ_2 = -2 of the second largest invariant of $\mathbf{S}^2 + \mathbf{\Omega}^2$ ($\mathbf{S}$ and $\mathbf{\Omega}$ being the symmetric and antisymmetric component of $\nabla \mathbf{u}$). This figure shows steady results obtained with two RANS models (k-ω SST and EARSM) and a flow field obtained by averaging instantaneous fields coming from unsteady DES computations, all computed on the same grid. RANS models yield tip vortices but they vanish more or less rapidly in the wake depending on the turbulence model used and the level of anisotropy associated with the turbulence model. With the DES approach, the tip vortices are maintained much further in the wake. These differences are caused by too high a level of turbulent kinetic energy associated with RANS turbulence models, even in the wake of the hub, see figure 4.

5.2.2 J = 0.45

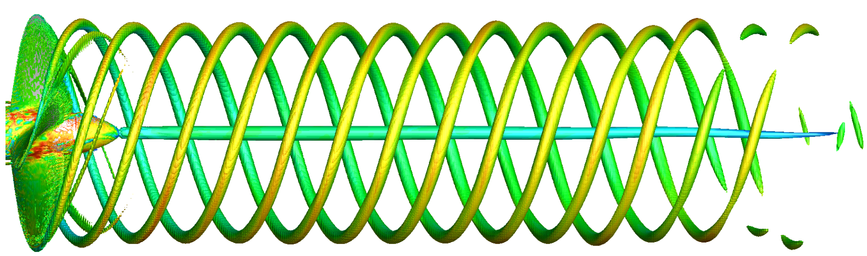
The comparison between RANS and DES simulations for the advance coefficient J = 0.45 is reported in figure 5. The characteristics of the flow are the faster deformation



(a) k-ω SST



(b) EARSM



(c) DES SST

Figure 3: J = 0.71 - Vortical structures visualizations (λ_2 = -2)

of the wake and the stronger tip vortices. Even if the tip vortices are stronger than in the previous case, they are resolved over a shorter distance. This trend is confirmed by the numerical results of Muscari et al. [6]. The DES permits to predict the vortices longer, and shows both the onset of the vortices instability and the start of the pairing process described in the experiments. It is clearly shown that the tip vortices deform from the helical path and tend to interact mutually and form a group. Then the hub vortex undergoes a sudden deformation from straight to a spiralling geometry.

6 CONCLUSIONS

The capabilities of numerical simulations with different turbulence models (RANS and DES) to predict the complex flow past an isolated propeller have been presented in this abstract. Two operational conditions have been considered: one for a moderate blade loading at J = 0.71, and a second for a higher blade loading at J = 0.45.

The RANS approach dissipates the tip vortices very quickly due to a high value of the turbulent kinetic energy in the vortices. The DES approach allows to capture the evolution of the tip vortices as long as the mesh is reasonably refined. The initial stages of the instability pattern, with two consecutive vortex filaments grouping their relative position, can also reproduced and follow with similarity the flow visualizations fro water channel experiments.

Acknowledgments

This work was granted access to HPC servers of CINES and IDRIS under the allocation 20129 made by GENCI (Grand Equipement National de Calcul Intensif) which is gratefully acknowledged.

REFERENCES

[1] Felli M., Camussi R. and Di Felice F. Mechanisms of evolution of the propeller wake in the transition and far fields. *J*ournal of Fluid Mechanics (2011) **682**:5–53.

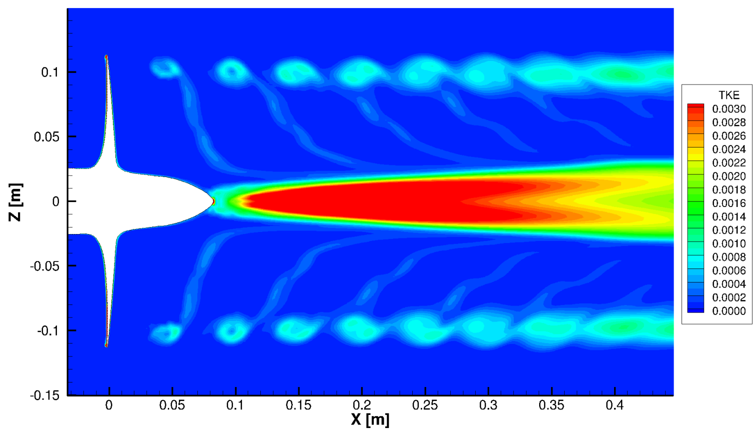
[2] Duvigneau R., Visonneau M. and Deng G.B. On the role played by turbulence closures in hull shape optimization at model and full scale. *J*ournal of Marine Science and Technology (2003) **8**:11–25.

[3] Queutey, P. and Visonneau, M. An interface capturing method for free-surface hydrodynamic flows. *C*omputers & Fluids (2007) **36**:1481–1510.

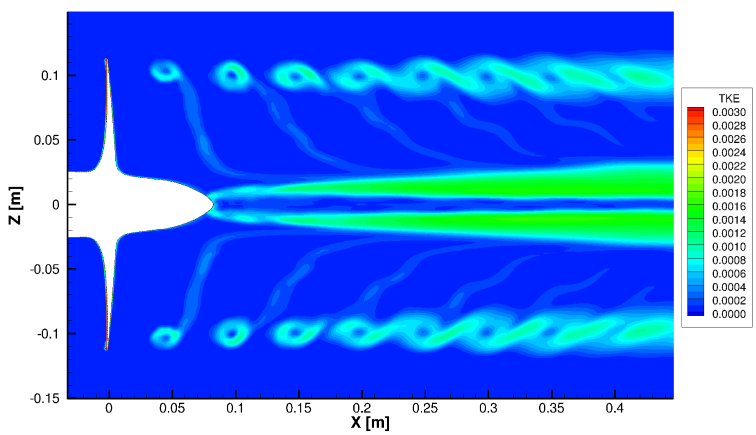
[4] Deng, G.B. and Visonneau, M. Comparison of explicit algebraic stress models and second-order turbulence closure for steady flows around ships. Proceedings of the 7th International Conference on Numerical Ship Hydrodynamics, Nantes, France, (1997).

[5] Guilmineau, E., Deng, G.B. and Wackers, J. Numerical simulation with DES approach for automotive flows. *J*ournal of Fluids and Structures (2011) **27**:807–816.

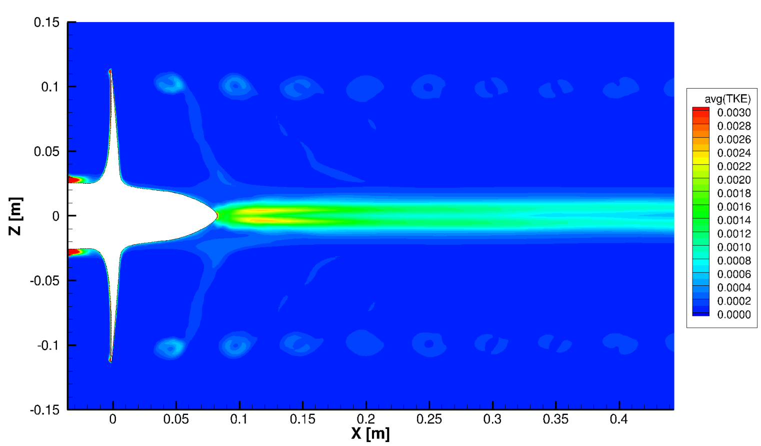
[6] Muscari R., Di Mascio A. and Vericco R. Modelling of vortex dynamics in the wake of a marine propeller. *C*omputers & Fluids (2013) **73**:65–79.



(a) k-ω SST

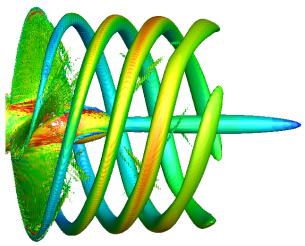


(b) EARSM

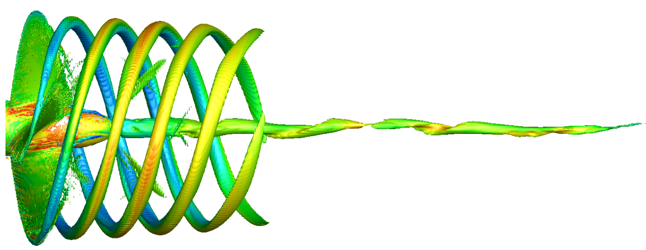


(c) DES SST

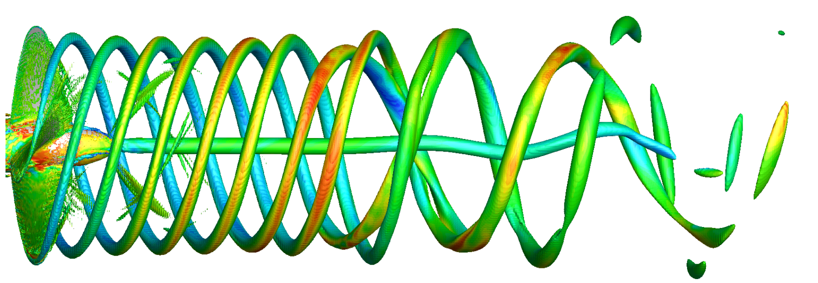
Figure 4: J = 0.71 - Turbulent kinetic energy



(a) k-ω SST



(b) EARSM



(c) DES SST

Figure 5: J = 0.45 - Vortical structures visualizations (λ_2 = -2)