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)
July 20 - 25, 2014, Barcelona, Spain

Viscous and turbulent modelling in a new Cartesian explicit solver for hydrodynamic applications

P. Bigay [1,2], G. Oger [1] , P-M. Guilcher [2] and D. Le Touzé [1]

[1] LUNAM Université, Ecole Centrale Nantes, LHEEA Lab. (UMR CNRS), Nantes, France
[2] HydrOcean, 1 rue de la Noë - BP32122, 44321 Nantes Cedex 3, France,

Key Words: *Explicit, Pseudo-compressible, Ghost-cell method, Viscosity, Turbulence*

CFD has now become an essential designing tool for engineers in offshore and naval hydrodynamic applications. Some relevant physical problems such as flows around ships with air entrapment in the bow jet, presence of bubbles in the wake, sea keeping of boats with complex moving appendages (thrusters ...) remain very difficult to simulate. One of the major drawbacks of CFD solvers is the important computational cost related to simulating accurately some complex configurations. The scheme presented in this paper aims at mitigating those issues and finding a compromise between accuracy, computational efficiency and easy handling of complex geometries. The chosen method is an Explicit Cartesian Finite Volume method for Hydrodynamics (ECFVH) based on a weakly compressible solver, with embedded or immersed interfaces. This explicit cell-centered resolution allows for an efficient parallel solving of very large simulations together with a straightforward handling of multi-physics. This method uses Adaptive Mesh Refinement so as to optimize the accuracy requirements together with the number of cells used. This paper focuses on the following features: the handling of complex geometries in a Cartesian framework, the treatment of viscosity and turbulence.

Firstly, the hyperbolic solver used for solving the Euler equations is described and its numerical dissipation effects are assessed. Complex boundary modeling is then addressed via two different methods: an embedded boundary method (cut-cell method) and an immersed boundary method (ghost-cell method). Viscous terms are adressed via different finite difference schemes (source-term or viscous fluxes). These solvers are being validated on academic test cases (Lid-Driven Cavity flow and 2-D flows over geometries). In the last part, the Large Eddy Simulation method is used to account for turbulent effects. Its validation and applicability is finally discussed.

REFERENCES

[1] C.Leroy, P.Bigay, G.Oger, D. Le Touzé, B.Alessandrini, *Preliminary developments of two-fluid Cartesian-grid Finite Volume Characteristic flux model for marine applications*. WCCM 2012 Proceedings.

[2] P.Bigay, C.Leroy, G.Oger, P.-M. Guilcher and D.Le Touzé, *Development of a new cartesian explicit solver for hydrodynamics flows*,Proceedings of the ASME 2013 32nd International Conference on Ocean, Offshore and Arctic Engineering (OMAE), 2013.