GFMFOAM: A VOLUME OF FLUID AND GHOST FLUID METHOD SOLVER FOR INCOMPRESSIBLE FREE SURFACE FLOW

PAULIN FERRO[1], PAUL LANDEL[2], MARC PESCHEUX[3]
[1]*SARL G-MET Technologies, paulin.ferro@g-met.fr*
[2]*SARL G-MET Technologies, paul.landel@g-met.fr*
[3]*SARL G-MET Technologies, marc.pescheux@g-met.fr*

Keywords: *VoF, GFM, free surface flows*

Incompressible two phases flows are widely used for ship and offshore hydrodynamic applications and still remain challenging to simulate properly. In this context two types of problems are generally encountered: transient simulations such as waves loads, seakeeping or any violent free surface flow and steady-state problems such as hydrodynamic resistances of ships/structures. While the first family of problems requires low Courant-Friedrichse-Lewy (CFL) numbers, the second one can accept high CFL numbers (> 100) in order to reduce the simulation time as long as the accuracy is preserved (which is a challenge for CFD codes). In a marine context, authors have traditionally reported the limitations of the standard incompressible two-phase flow solver interFoam. Firstly, Meyer et al. [1] have shown that MULES algorithm (Multidimensional Universal Limiter with Explicit Solution, [2]) needs additional iterations (PISO [3] and SIMPLE [4]) and sub-cycles for sufficient stability in comparison to a commercial solution for large time step simulations. Secondly, the presence of spurious air velocities has been widely reported ([5], [6], [7]). Vukčević et al. [6] have stated that spurious air velocities occur due to the imbalance between pressure and density gradients during the pressure-velocity coupling and propose the use of the Ghost Fluid Method (GFM) [8] to solve this issue. In this context, an incompressible two-phase flow solver (named gfmFoam) based on Volume-of-Fuid [9] and GFM is proposed. Firstly, in gfmFoam, the convection term in the phase fraction equation is discretized using algebraic schemes: CICSAM [10], MCICSAM [11], HRIC [9], MHRIC [13] or BICS [14] unlike the standard approach : MULES. The deferred correction method is retained to increase numerical stability. Secondly, gfmFoam benefits of the GFM for pressure and density discontinuity handling (Figure 1). As indicated above, the use of GFM is motivated by its ability to alleviate light phase acceleration and preserve a continuous velocity field across the free surface. This results in lower CFL number allowing larger time step. The method, implemented following [6] has a different momentum formulation [15]. Finally, gfmFoam is tested on various test cases ranging from dam break transients to ship resistance assessment. Comparisons with interFoam solver are carried out in term of accuracy, robustness and CPU time.

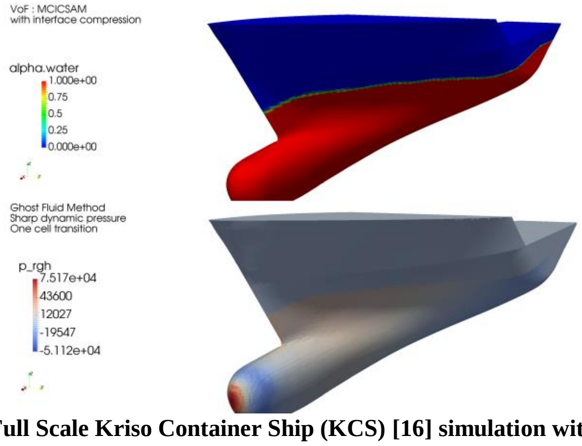

Figure 1: Full Scale Kriso Container Ship (KCS) [16] simulation with gfmFoam

Acknowledgements

The authors thank all those involved in the organisation of OFW16 and to all the contributors that will enrich this event.

References

[1] Meyer, J., Renzsch, H., Graf, K. and Slawig, T., Advanced CFD-Simulations of free-surface flows around modern sailing yachts using a newly developed OpenFOAM solver, The 22nd Chesapeake Sailing Yacht Symposium, Annapolis-Maryland, (2016).

[2] Damián, S.M., Nigro, N.M., 2014. An extended mixture model for the simultaneous treatment of small-scale and large-scale interfaces. Int. J. Numer. Methods Fluid. 75 (8), 547–574.

[3] R.I Issa, Solution of the implicitly discretised fluid flow equations by operator-splitting, Journal of Computational Physics, Volume 62, Issue 1, 1986, Pages 40-65, ISSN 0021-9991.

[4] S.V Patankar, D.B Spalding, A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows, International Journal of Heat and Mass Transfer, Volume 15, Issue 10, 1972, Pages 1787-1806, ISSN 0017-9310.

[5] Kim, G.-H., Park, S., 2017. Development of a numerical simulation tool for efficient and robust prediction of ship resistance. Int. J. Naval Architec. Ocean Eng. 9 (5), 537e551.

[6] Vukčević, V., Jasak, H., Gatin, I., 2017. Implementation of the ghost fluid method for free surface flows in polyhedral finite volume framework. Comput. Fluids 153, 1–19.

[7] Larsen BE, Fuhrman DR, Roenby J (2019) Performance of interFoam on the simulation of progressive waves. Coast Eng J.

[8] Fedkiw, R.P., Aslam, T., Merriman, B., Osher, S., 1999. A non-oscillatory eulerian approach to interfaces in multimaterial flows (the ghost fluid method). J. Comput. Phys. 152 (2), 457–492.

[9] Hirt, C., Nichols, B., 1981. Volume of fluid (VOF) method for the dynamics of free boundaries. J. Comput. Phys. 39 (1), 201–225.

[10] Ubbink O., Issa R.I., 1999, Method for capturing sharp fluid interfaces on arbitrary meshes, J. Comput. Phys., 153, 26-50.

[11] Wacławczyk, T., 2007, Novel high-resolution scheme for interface capturing in multi-phase flow, 6 th International Conference on Multiphase Flow, ICMF 2007, Leipzig, Germany, July 9 – 13, 2007.

[12] Muzaferija S., Peric M., Sames P., Schelin T., 1998, A two-fluid NavierStokes solver to simulate water entry, Proc. Twenty-Second Symposium on Naval Hydrodynamics.

[13] 2009, Fluent., Fluent 12.0 Users Guide. Fluent Inc.

[14] Queutey, P., Visonneau, M., 2007. An interface capturing method for free-surface hydrodynamic flows. Comput. Fluids 36 (9), 1481–1510.

[15] Petteri Peltonen, Pekka Kanninen, Erkki Laurila, Ville Vuorinen, The ghost fluid method for OpenFOAM: A comparative study in marine context, Ocean Engineering, Volume 216, 2020, 108007, ISSN 0029-8018.

[16] Kim, W.J., Van, S.H., Kim, D.H., 2001. Measurement of flows around modern commercial ship models. Exp. Fluid 31 (5), 567e578.