Using grid adaptation to understand ship flow instability

Jeroen Wackers, Emmanuel Guilmineau and Michel Visonneau

LHEEA Laboratory, UMR CNRS 6598, Ecole Centrale de Nantes, France.
e-mail: jeroen.wackers@ec-nantes.fr

ABSTRACT

One of the reasons for developing and using adaptive grid refinement in the simulation of fluid flow is that the locally fine grid resolution can reveal details of the flow which cannot be observed on non-adapted grids. Thus, grid adaptation can lead to new discoveries about the flow physics. However, before presenting such a discovery, one needs to be sure that the observed flow behaviour is not due to perturbations coming from the adaptation itself.

Recently, using the adaptive refinement in the flow solver ISIS-CFD developed by our group [1], we observed unsteady flow behaviour for Reynolds-averaged Navier Stokes (RANS) simulations of two standard ship flow test cases, which are in general thought to be steady. The first of these is the flow around the KVLCC2 tanker ship. This case was introduced about 15 years ago, a recent study by several research teams using different flow solvers [2] yielded steady results for all RANS computations. Our simulations, on the contrary, contain a small zone of unsteady vortex separation on the lower hull just in front of the ship's propeller.

The Japan Bulk Carrier (JBC) test case was created for the recent Tokyo 2015 workshop [3]. Using RANS, we find unsteady vortex shedding on the sides of the skeg which supports the propeller. Such unsteady behaviour has also been observed on non-adapted grids if Detached Eddy Simulation (DES) is used [4]. All published RANS simulations are steady.

The goal of this paper is to determine whether the flow analysis using adapted grids is valid, or if the unsteadiness observed comes from numerical artifacts, notably the repeated modification of the mesh during the simulation which is inherent in our our grid adaptation technique. Therefore, the paper looks at the factors which influence the flow (un)steadiness. Specifically, if a flow is physically unsteady but yields a steady simulation result, it is forced to steadiness by (a) numerical diffusion, (b) a steady flow solver, or (c) forced symmetry through a vertical symmetry plane. On the contrary, if a flow is physically steady but its simulation is unsteady, then spurious energy is added to the flow numerically. A probable source of this energy would be (d) adaptive grid refinement.

These four aspects are tested through systematic variation of the simulation parameters. Numerical diffusion is changed by varying the mesh size of the adapted grid, results using steady and unsteady solvers are compared, as well as simulations with or without symmetry planes. Finally, the effect of grid adaptation is assessed by running full simulations on the final grid of the adaptation process and on non-adapted grids where fine cells have been inserted locally around the regions of instability. If these studies show the validity of the unsteady ship flows observed, this would be an important conclusion for ship hydrodynamics.

REFERENCES

[1] J. Wackers, A. Leroyer, G. B. Deng, P. Queutey, M. Visonneau, “Adaptive grid refinement for hydrodynamic flows”, *Comput Fluids*, Vol. **55**, pp. 85-100, (2012).

[2] L. Larsson, F. Stern, M. Visonneau, *Numerical Ship Hydrodynamics, An assessment of the Gothenburg 2010 Workshop*, Springer (2014).

[3] M. Visonneau, “JBC local flow analysis”, *Tokyo 2015 Workshop on CFD in Ship Hydrodynamics*, http://www.t2015.nmri.go.jp/Presentations/Day1-PM1-JBC-LocalFlow-Visonneau.pdf (2015).

[4] M. Visonneau, G.B. Deng, E. Guilmineau, P. Queutey, J. Wackers, “Local and Global Assessment of the Flow around the Japanese Bulk Carrier with and without Energy Saving Devices at Model and Full Scale”, *31st Symposium on Naval Hydrodynamics*. Monterey, California (2016).