Simulation of cavitation and surface erosion on marine propellers

HORIZON2020
Marie Skłodowska-Curie actions

Abstract

The prediction of the wakefield behind the hull is considered as the first important step for the evaluation of the propeller performance in a cavitating flow.

In this study the commercial CFD software StarCCM+ is applied as a RANS solver for the prediction of the resistance and the nominal wakefield behind a 4800 TEU Container Vessel in calm water and Froude number 0.2182. Different turbulence models are examined for fully turbulent flow. Additionally, laminar flow has been examined as well. The axial velocities on the propeller plane are predicted and compared to experimental data.

The underprediction of the velocities in the fully turbulent flow and the overprediction in the laminar flow has led to the use of a transition model. The transition model available in StarCCM+ is the Gamma ReTheta transition model, a correlation-based model designed for implementation in modern unstructured, parallel CFD codes.

The γ- Re_θ transition model requires the first computational node of the grid to be located at y^+ = 0.1-1.0 from the wall. To this end, all the other turbulence models were used with both high and low y^+ wall treatment. It is demonstrated that the predicted resistance with the standard k-ε turbulence model is the closest to the obtained value, however the transition model has the best prediction for the axial velocities in the wakefield.

Introduction

In this research project the primary objective is to develop a CFD methodology capable of detecting the time dependent development of the cavities on the propeller blades and thereafter evaluating the risk of erosion on the surface of the propeller. Industrial applications require accurate results in an acceptable time span. For that reason, at the moment the faster and less power and time consuming RANS are considered as the best option for industrial use and performance.

The functioning of a propeller in a wakefield and the pressure pulses induced to the hull by a cavitating propeller have to be predicted accurately enough. The prediction of a ship's nominal wake-field is important since it provides a good initial estimate for the RANS propeller model. As a result the prediction of the nominal wake by evaluating the performance of different turbulence models is at the forefront during the first stage of the project.

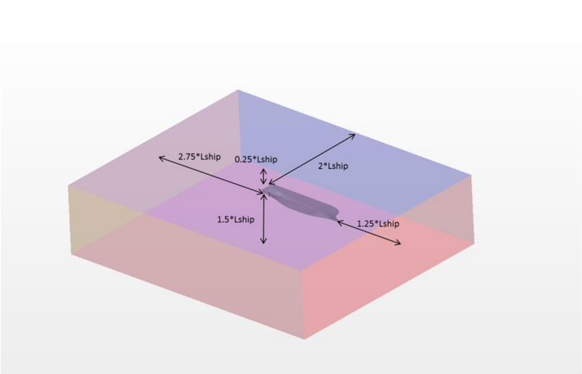

Figure 1. Dimensions of the domain.

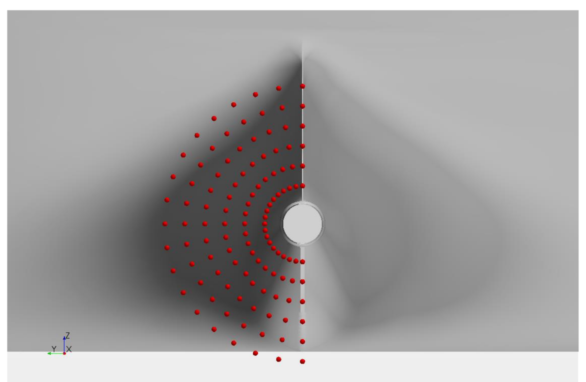

Figure 2. Points on the propeller plane where the axial velocities are calculated and monitored.

Methods and Materials

All computations reported here were performed using half the geometry in order to reduce the number of cells and therefore the computation time. The dimensions of the full domain are shown in fig. 1.

In addition to the symmetry plane of the hull, the lateral boundary parallel to it (outer) was considered as a slip wall (the same for the top boundary). At the downstream boundary (outlet), hydrostatic pressure corresponding to the undisturbed water surface was prescribed. Upstream and bottom boundaries were treated as inlets with prescribed velocity (1.833 m/s) and volume fraction. A wave damping option for the inlet, the outlet and the outer boundaries was activated to avoid radiation waves.

The same grid topology was used for every computation, namely, trimmed hexahedral grids with local refinements and prism layers along walls. Two cases were examined for each turbulence model, one with high y^+ at near-wall cells (y^+ >30) using the wall-function-type approach and one with low y^+ (y^+ <1) to resolve the viscous sublayer. In order to avoid using fine grid in areas where it is not necessary, local volumes of different shape were created resulting in mesh structure shown in Fig. 3.

Free surface is modeled using Volume-of-Fluid approach and a high resolution interface-capturing scheme. VOF waves were used to simulate surface gravity waves on the light fluid-heavy fluid interface. In this case a flat wave has been chosen to represent the calm plane of the water.

At the beginning, all the calculations have been done with the hull in a fixed position with the same trim and sinkage as measured in the model test for this speed (0.1 deg trim by stem, 0.00984 m sinkage). At the next step the computations have been done with the hull in its floating position (even keel) and Dynamic Fluid Body Interaction (DFBI) is applied to simulate the motion of the vessel.

In order to cope with any velocity fluctuations each and every point on the propeller plane has been monitored separately (Fig. 2) and the mean value of the last ten seconds has been taken. The time step used for the computations was calculated in reference to the ship's length and speed and is 0.47.

Results

The percentage difference between the experimental obtained values and the predicted values for the axial velocities on the propeller plane has been calculated for each point and then the mean percentage value of the whole propeller plane has been taken.

The results with the hull in the fixed position showed that the predicted values when using wall functions where closer to the experimentally obtained values than the predicted values with low y+ wall treatment. As a result the DFBI was performed only to the meshes with high y+ values. Thus, table 1 shows only the results for the transition model and the rest turbulence models using DFBI.

None of the turbulence models could predict accurately enough the velocity distribution very close to the shaft (more than 5% difference). Nevertheless, the most critical area for cavitation inception is around the propeller tip (tip vortex cavitation) and mostly the top area of the propeller blades (sheet cavitation, cloud cavitation etc.). To this end only the results regarding relative radii r/R greater than 0.5 and 0.7 (which are of the highest importance) are being presented.

Table 1. Hull resistance and mean percentage difference of the predicted axial velocities to the obtained values on the propeller plane.

Turbulence Model	Resistance (N)	Avg. Above r/R=0.5	Avg. Above r/R=0.7
laminar	24,38 (-56,17%)	14,28%	10,19%
standard k-ε high y+	55,81 (+0,33%)	5,61%	5,14%
realizable k-ε high y+	54,71 (-1,66%)	5,07%	4,55%
k-ω SST high y+	53,18(-4,41%)	4,89%	4,22%
k-ω γ-Re_θ	38,07(-31,56%)	4,51%	2,59%

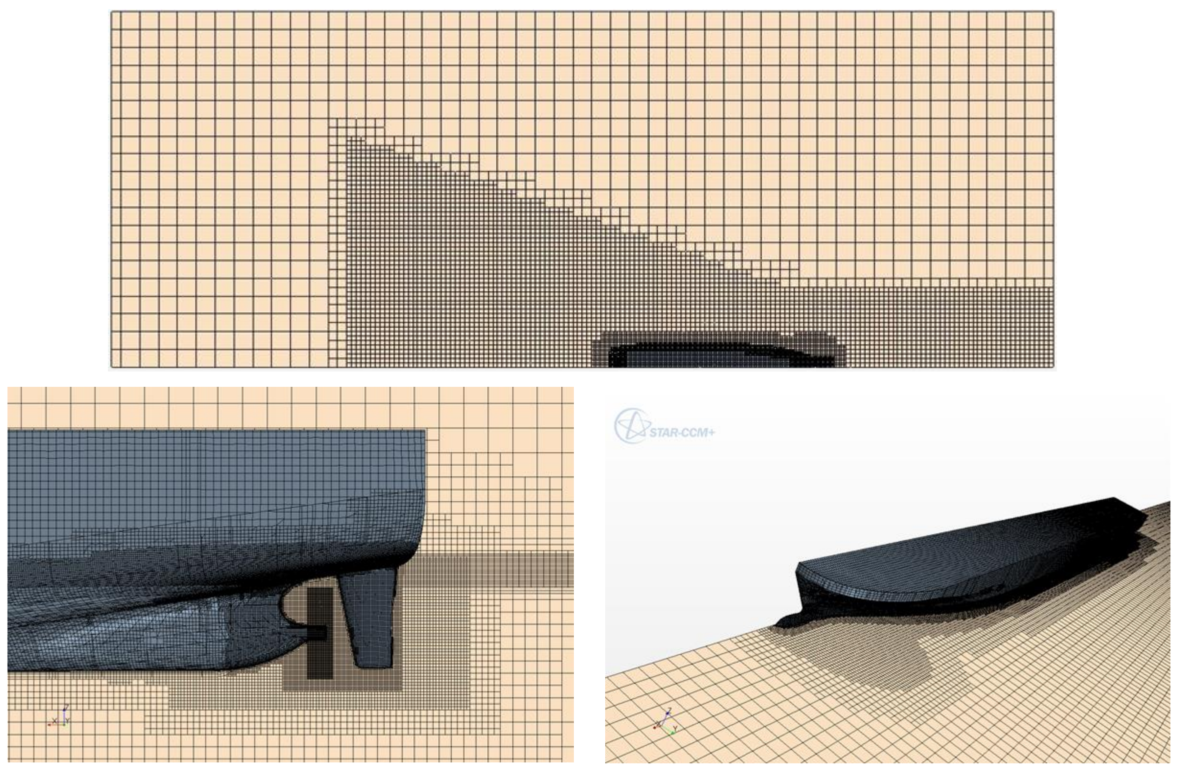

Fig. 3. The structure of the grid around the hull, showing the local refinements for the bow, the wake, the free surface and the wave zone.

Discussion

The results have shown that in fully turbulent flow the predicted axial velocities are a little lower than the experimentally obtained values. On the contrary, in laminar flow they are quite higher. This is happening because when the flow is laminar then the separation of the flow is taking place very late and therefore the wake of the flow at the propeller plane covers only a small area. In general, the smaller the wake the higher the velocities because they are influenced more by the freestream. With the γ-Re_θ transition model the flow separation point moves backwards with respect to the fully turbulent case giving higher velocities on the propeller plane than the fully turbulent flow and getting closer to the measured values.

Conclusions

The presented results show that one can obtain a reliable prediction of the wakefield of container vessels using the γ-Re_θ transition model. Although the predictions very close to the propeller shaft are not accurate enough, this is not a critical area. On the other hand, the predicted resistance is significantly lower than the measured value. Since the current research work focusses on the dynamics of the cavities, the transition model is regarded to be valid for further work.

Contact

Themistoklis Melissaris
Wartsila Netherlands B.V.
Email: themis.melissaris@wartsila.com
Website: cafe-project.eu
Phone: +31 616201532

References

1. Wilcox, D.C. (2006): "*Turbulence Modeling for CFD*", 3rd Edition, DCW Industries, California.
2. Ferzinger, J.H. abd Peric, M. (2003): "Computational Methods for Fluid Dynamics", 3rd Edition, Springer Verlag, Berlin, Heidelberg.
3. Schobeiri, M.T., (2010): "Fluid Mechanics for Engineers, A Graduate Textbook", Springer Verlag, Berlin, Heidelberg.
4. Patankar, S.V. (1980): "Numerical Heat Transfer and Fluid Flow", Hemisphere Publishing Corporation, US.
5. Versteeg, H.K. and Malalasekera, W. (1995): "An Introduction to Computational Fluid Dynamics, The Finite Volume Method", Longman Group Ltd, US.
6. Sodja, J. (2007): "Turbulence models in CFD", University of Ljubljana, Faculty for mathematics and physics, Department of physics.
7. Enger, S., Peric, M. and Peric, R. (2010): "Simulation of Flow Around KCS-Hull", CD-adapco, A workshop on CFD in Ship Hydrodynamics, Gothenburg, Sweden.
8. Mala, P., Suluksna, K. and Juntasaro, E. (2009): "Calibrating the γ-Re_θ Transition Model for Commercial CFD", 47th AIAA Aerospace Sciences Meeting, Florida.
9. Tsaggaris, S. (2005): "Fluid Mechanics", Symewn Publishing Corporation, Athens, Greece.