Fourth International Symposium on Marine Propulsors
smp'15, Austin, Texas, USA, June 2015

Investigation of Azipod thruster with nozzle performance by CFD simulations and experiments

Pasi Miettinen, Mikael Lindfors

ABB Marine
Helsinki, Finland

ABSTRACT

ABB Marine is using Computational Fluid Dynamics (CFD) simulations for improving hydrodynamic efficiency of Azipod thrusters. Open water performance of pushing type Azipod thruster with nozzle is presented. Unsteady RANS CFD simulations by sliding mesh approach were performed at full and model scales and compared to experimental model tests for validation. In this study, the thruster is tilted and SST (Menter) k-ω turbulence model and commercial CFD code StarCCM+ version 9.02.005 is used. Simulations capture the shape of open water curves (propeller thrust, propeller torque, nozzle thrust and total thrust) very well at both scales. Discrepancies between simulations and experiments are discussed and differences between full and model scales are addressed.

Keywords

CFD, RANS, ducted propeller, transient simulation

1 INTRODUCTION

Reynolds-Averaged Navier Stokes (RANS) approach has become the standard in industrial CFD simulations due to its accuracy and simulation speed, especially for steady state simulations. For transient simulations RANS is also the most probable choice but increased computing power and accuracy requirements may lead to more advanced turbulence models such as Detached Eddy Simulations (DES) or even Large Eddy Simulations (LES). In marine industry and especially for propeller and propeller-pod simulations RANS is widely used with either quasi-steady Moving Reference Frame (MRF) or unsteady sliding mesh approaches (Moustafa, A-M. et al. 2011). Variants of k-epsilon and k-ω turbulence models are mainly utilized in the simulations.

At ABB Marine RANS is the main CFD approach with the unsteady sliding method since CFD is not only used for open water performance but also for estimating hydrodynamics loads at varied nonzero steering angles. Eddy viscosity model is most of the time applied in the simulations but most complicated RANS approach, namely Reynolds Stress Transport (RST) has been initially tested for some simulation cases. Detached Eddy Simulation (DES) has also been tested for cases where pure RANS shows features of instabilities due to significant turbulence. However, in this paper RANS approach is adopted for simulations of open water performance of Azipod thruster with nozzle. Hydrodynamic performance is simulated at model and full scales and compared to experimental model tests for validation.

2 FORMULATION

2.1 Unsteady RANS simulation at full and model scales

Unsteady Reynolds-Averaged Navier Stokes (URANS) approach with SST (Menter) k-ω turbulence model is used for simulating pushing type Azipod thruster. Commercial CFD code StarCCM+ version 9.02.005 is used. In this study the tilted thruster is equipped with the nozzle as illustrated in Fig. 1. The simulation geometry exceeds 10 propeller diameters upstream from the nozzle and 20 propeller diameters downstream from the torpedo. In the vertical direction, the simulation domain exceeds 10 propeller diameters above the strut and below the nozzle and in the lateral direction 20 propeller diameters from the nozzle. Simulation domain is divided into rotating and stationary regions. The rotating region includes propeller blades, blade tips and hub and the stationary region includes strut, nozzle, supports and torpedo together with inlet, outlet and sides surfaces of the simulation domain. The inlet surface is described as velocity inlet, the outlet surface as pressure outlet and side surfaces are defined as symmetry planes.

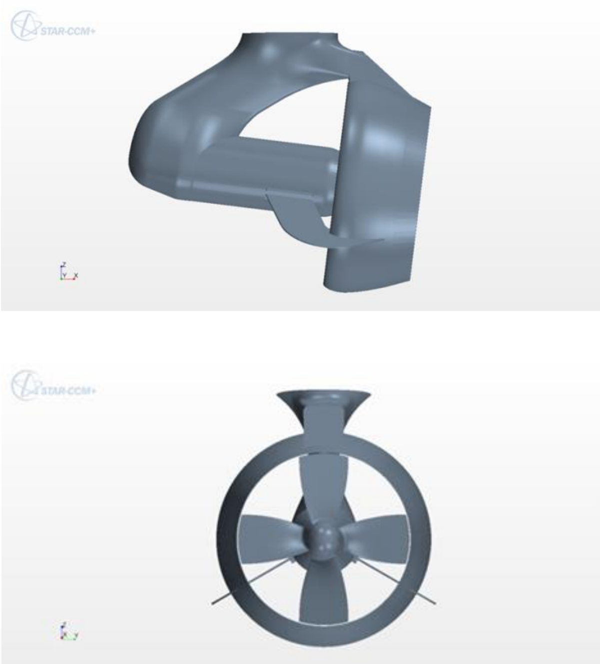

Figure 1: Side and back views of the geometry with pod, the nozzle, supports and the propeller.

The rotating and stationary regions are meshed with trimmer cells and prism layers. The total number of cells is 26.4 million and the same mesh was used at all simulated inflow velocities. The mesh was generated at full scale and the resulting volume mesh was scaled for model scale simulations. Second-order discretization schemes were used for mass, momentum and turbulent equations. Propeller blade diameter is D = 3.6 m at full scale and 0.25 m at model scale. As the sliding mesh approach is used, the time step size was set to 1 degree/time step. At inlet velocity of 15 knots at full scale, the average wall distance is $Y^+ = 78$ and at advance number of 0.1 at model scale, the average wall distance is $Y^+ = 1.2$. Figs. 2-3 show mesh resolutions on the propeller surface and on the gap between the nozzle and propeller blade tips.

Figure 2: Mesh resolution on the propeller surface.

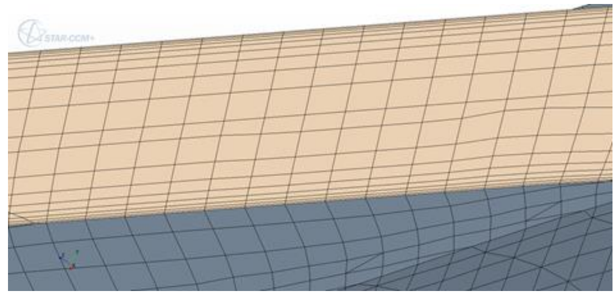

Figure 3: Mesh resolution on the gap between the nozzle and propeller blade tips.

3 RESULTS AND DISCUSSION

3.1 Open water performance at model scale

Azipod thruster with the nozzle is tested at model scale at the external model basin. Fig. 4 shows open water performance for the propeller, the nozzle and the pod unit. URANS simulations at model scale are compared to these experiments. In general, simulations capture the shape of curves very well, especially for propeller thrust, propeller torque, nozzle thrust and pod unit thrust. The best match between simulations and experiments is observed for the nozzle thrust and propeller torque, but propeller and pod unit thrust are clearly underestimated by the simulations. In Fig. 5 water velocity magnitude is illustrated at the plane of the propeller axis at model scale.

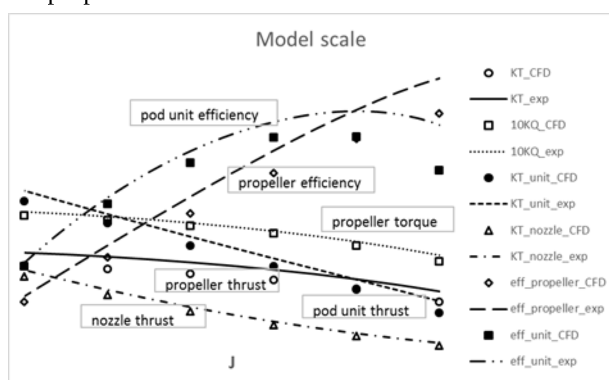

Figure 4: Open water performance at model scale for the propeller, the nozzle and the pod unit. Symbols are CFD results at model scale.

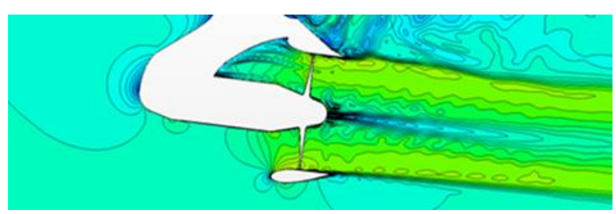

Figure 5: Water velocity magnitude at the plane of the

propeller axis at model scale.

3.2 Open water performance at full scale

Fig. 6 shows simulated open water performance at full scale for the propeller, the nozzle and the pod unit. Simulated behavior is compared to model scale experiments. In general, simulations capture the shape of curves very well for each of the parameters including propeller and pod unit efficiencies. This is very interesting result since only pod unit thrust (and therefore pod unit efficiency) is scaled from model scale experiments. The agreement is pretty good between simulations and experiments for the pod unit thrust, propeller and pod unit efficiencies and also for the nozzle thrust. The differences between full and model scales are addressed in the next section. In the Fig. 7 water velocity magnitude is illustrated at plane of the propeller axis at full scale. As expected velocity field appears to be more turbulent at full scale.

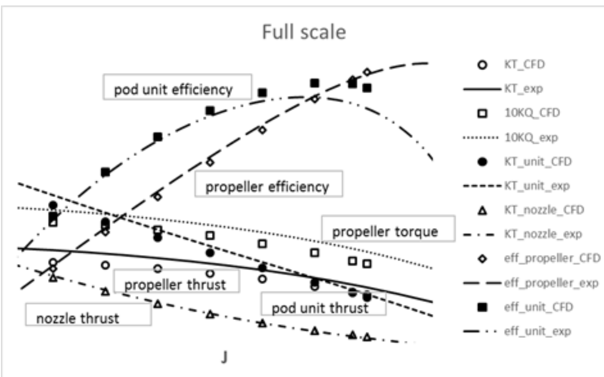

Figure 6: Open water performance at full scale for the propeller, the nozzle and the pod unit. Symbols are CFD results at full scale.

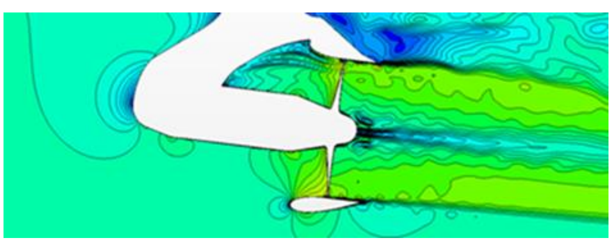

Figure 7: Water velocity magnitude at the plane of the propeller axis at full scale.

3.3 Comparison between model and full scales

Fig. 8 shows comparison between simulated open water performance at full and model scales. At full scale open water efficiency is higher as expected due to Reynolds scaling and as reported by Bulten and Suijkerbuijk (Bulten and Suijkerbuijk 2013). Nozzle and propeller thrust are very similar at both scales. Propeller torque is smaller and pod unit thrust is slightly larger at full scale. All these findings are similar to the observations by Bulten and Nijland (Bulten and Nijland 2011). They contribute propeller torque and thrust behavior to Reynolds scaling as observed for open propellers. Propeller torque will decrease and thrust will increase. However, for ducted propellers lower viscous losses will increase the flow rate through the nozzle at full scale which results in smaller propeller thrust and torque.

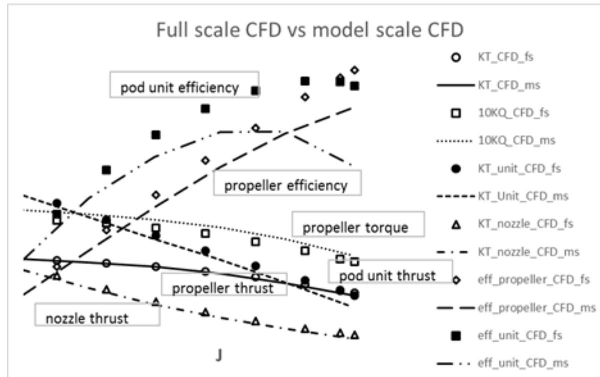

Figure 8: Comparison between full and model scale CFD. Symbols are CFD results at full scale.

3.4 Importance of unsteady analysis

Fig. 9 shows the difference between vorticities downstream of the Azipod thruster simulated by Moving Reference Frame (MRF) (left) or Sliding Mesh (right) approaches. Propeller hub generated streamlines are also plotted. It is evident that MRF cannot capture fluid vorticity correctly since propeller blade positions are clearly visible in vorticity field. This is due to the fact that propeller blades are not actually rotating in this numerical approach. The solution for this would be simulations at varied propeller blade positions and then calculate the average vorticity. However, even then this quasi-steady approach cannot produce dynamic effects such as propeller blade passage or interactions between propeller, nozzle and pod. Figs. 10-11 illustrate simulated propeller thrust, torque, nozzle and pod unit thrusts during one propeller revolution. Periodicity of forces/moment is obvious and it is related to the number of propeller blades (four). Propeller-nozzle or nozzle-pod or propeller-pod interactions generate other peaks in forces/moments.

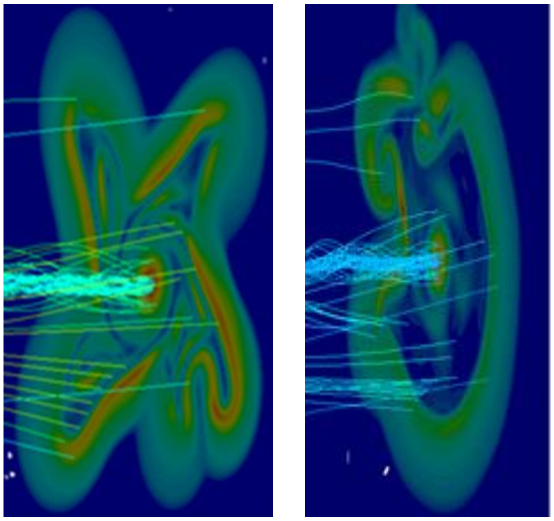

Figure 9: Comparison between vorticities at the plane downstream of Azipod thruster. Simulations by Moving Reference Frame (left) or Sliding Mesh (right) approaches. Propeller hub generated streamlines are also illustrated.

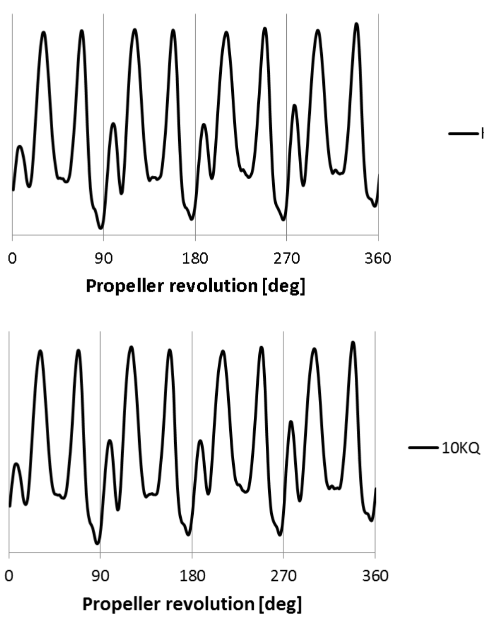

Figure 10: Simulated propeller thrust and torque variation during one propeller revolution at full scale.

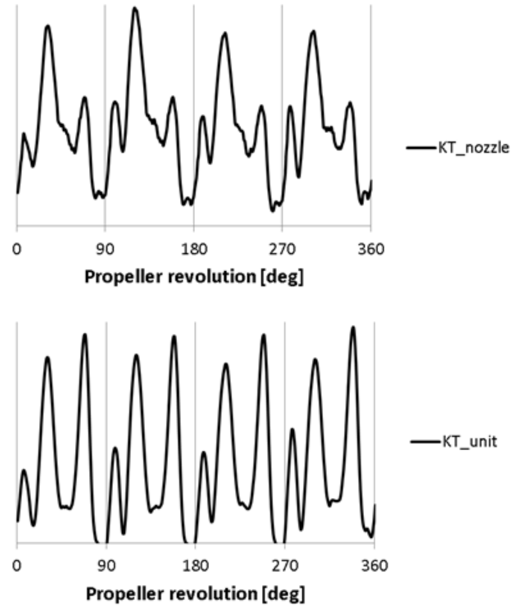

Figure 11: Simulated nozzle thrust and pod unit thrust variation during one propeller revolution at full scale.

4 CONCLUSIONS AND FUTURE WORK

Open water performance of tilted Azipod thruster with nozzle is simulated using Computational Fluid Dynamics (CFD). Unsteady simulations were performed at full and model scales and compared to experimental model tests performed at the external model basin. In general, simulations capture the shape of open water curves very well at full and model scales but simulated values are lower than measured ones, especially propeller thrust is too small at both scales. There is a constant difference between simulated and measured propeller thrust. It could be that a small part of the set up or boundary condition information is missing in the CFD model compared to the test set up.

It might be that RANS model with eddy viscosity model is insufficient for capturing all the phenomena occurring near the propeller and the nozzle regions at model scale and more advanced yet more complicated (Reynolds Stress Transport (RST) or Detached Eddy Simulations (DES)) models should be utilized in the future. RST approach accounts for turbulence anisotropy due to strong swirl motion, streamline curvature and rapid changes in the strain rate (StarCCM+ UserGuide 9.02). DES approach uses RANS approach at boundary layers and irrotational flow regions and large eddy simulation (LES) approach for unsteady separated regions if the simulation mesh is fine enough. Or perhaps measurement inaccuracies for tilted thruster are partially responsible for the gap between simulations and measurements.

ACKNOWLEDGMENTS

Support from colleagues at ABB Marine and from CD-adapco is fully acknowledged. Gridcore cluster is utilized in this study.

REFERENCES

Moustafa, A-M. et al. (2011), “Workshop on Cavitation and Propeller Performance,” Proc. of 2nd International Symposium on Marine Propulsors, Hamburg, Germany.

Bulten, N.W.H. and Suijkerbuijk, R. (2013), “Full scale thruster performance and load determination based on numerical simulations,” Proc. of 3rd International Symposium on Marine Propulsors, Tasmania, Australia

Bulten, N.W.H. and Nijland, M. (2011), “On the development of a full-scale numerical towing tank,” Proc. of 2nd International Symposium on Marine Propulsors, Hamburg, Germany.

StarCCM+ UserGuide 9.02. CD-adapco 2014.