CHALMERS
UNIVERSITY OF TECHNOLOGY

CFD Simulations of the Japan Bulk Carrier Test Case

Downloaded from: https://research.chalmers.se, 2025-04-18 06:51 UTC

Citation for the original published paper (version of record):
Andersson, J., Hyensjö, M., Eslamdoost, A. et al (2015). CFD Simulations of the Japan Bulk Carrier Test Case. NUMERICAL TOWING TANK SYMPOSIUM. 18TH 2015. (NUTTS 2015)

N.B. When citing this work, cite the original published paper.

research.chalmers.se offers the possibility of retrieving research publications produced at Chalmers University of Technology. It covers all kind of research output: articles, dissertations, conference papers, reports etc. since 2004. research.chalmers.se is administrated and maintained by Chalmers Library

(article starts on next page)

CFD Simulations of the Japan Bulk Carrier Test Case

Jennie Andersson[1], Marko Hyensjö[2], Arash Eslamdoost & Rickard Bensow

Department of Shipping and Marine Technology, Chalmers University of Technology, Gothenburg, Sweden
[2]Rolls-Royce Hydrodymanic Research Centre, Kristinehamn, Sweden
[1]jennie.andersson@chalmers.se

This computational fluid dynamics (CFD) validation study is performed as a foundation for further studies with focus on the interaction effects between propulsor and hull. To be able to study the interaction effects, an appropriate CFD methodology need to be established and validated for a bare hull, for the propulsion unit and for the combined system, a self-propelled hull.

The work to validate a CFD model is initiated through the use of the JBC, Japan Bulk Carrier, open test case. The JBC test case is developed for the 2015 workshop on CFD in Ship Hydrodynamics. The tested JBC only exists in model scale with scale factor 1:40 (L_{PP} = 7 m). Model ship speed is 1.179 m/s, corresponding to Fn = 0.142 and 14.5 kn, only calm water conditions are tested. There are two variants of the hull, with and without an energy saving device, within this study the one without is used. Test data used for validation of the CFD results are from towing tank experiments at NMRI. The aim of further studies is to study propulsor hull interaction in full scale, but since detailed test data in full scale is limited, all computations will be performed in model scale.

The commercial CFD package STAR-CCM+, a finite volume method solver, is employed for all studies. STAR-CCM+ is a general purpose CFD code used for a wide variety of applications. It solves the conservation equations for momentum and mass, turbulence quantities and volume fraction of water using a segregated solver based on the SIMPLE-algorithm. A 2nd order upwind discretization scheme in space is used. It is of interest to study how a general purpose code can perform for detailed ship hydrodynamic analyses and which limitations that could be identified.

1. Bare hull simulation

For this case, simulations are performed with free surface and the hull free to heave and pitch, but only with a half hull since symmetrical conditions are assumed. Focus has been on construction of a computational grid and selection of turbulence model, suitable for the purpose of wave pattern prediction, detailed studies of the flow field at the stern and overall resistance prediction.

The free surface is modelled using the Volume-of-fluid (VOF) method, implying that the domain is consisting of one fluid whose properties varies according to the volume fraction of water/air. The convective term is discretized using the High resolution interface capturing (HRIC) scheme [1]. The heave and pitch motion is modelled with the *DFBI Equilibrium* model, implemented in STAR-CCM+ v10.4; the model moves the body stepwise to obtain balanced forces and moments. Initially, effort was put into obtaining results using a model where the motion of the hull is physically correct modelled (*DFBI Free motion*). However, satisfying results could not be obtained using this model due to large oscillations in the results, similar problems have earlier been noticed [2]. Since a steady state solution is sought and time accuracy is not of importance, a 1st order implicit scheme is used for time integration, the time step is 0.04s.

The domain size in [X,Y,Z] is given by [$-3.5L_{PP}$: $2.5L_{PP}$, 0:$2L_{PP}$, $-1.5L_{PP}$:$1L_{PP}$] and where L_{PP} is the length between perpendiculars for the model ship, [0,0,0] at mid-ship. An inlet velocity boundary condition is specified at the inlet, upper, lower and outer sides of the domain. At the outlet a static pressure is specified. The water surface level is initialized as the declared draft of the hull (0.4125 m above the keel). A symmetry plane is specified at mid-ship.

The computational grids are generated using STAR-CCM+. A trimmed hexahedral grid (polyhedral cells) with prism layers along the walls is used. A mesh sensitivity study consisting of three grids is performed and the same mesh topology is applied for all grids and the level of refinement is modified through a characteristic cell size. Volumetric controls and anisotropic refinements are used to refine the region close to the hull, the wake and the water surface in general, see Figure 1 for the resulting mesh structure.

Within this study, a comparison is made between resolved boundary layers down to y+ < 1 and application of wall functions. For the resolved boundary layers (denoted y+ < 1) 31 prism layers with a near wall cell height of 0.02 mm is applied. When applying a wall function model 8 prism layers are used, with a near wall cell height of 4.4 mm, corresponding to y+ > 30 over the hull surface. The prism layers are kept constant in height for the different mesh refinements. The coarse, medium and fine grids had $3.3 \cdot 10^6$, $4.8 \cdot 10^6$ and $6.8 \cdot 10^6$ cells respectively for y+ > 30, and $7.1 \cdot 10^6$, $9.9 \cdot 10^6$ and $13.9 \cdot 10^6$ for y+ < 1, implying a ~100 % increase in mesh count when resolving the boundary layers.

Suitable turbulence models, for the investigation of ship hydrodynamics are thoroughly discussed in [3]. The main conclusion is that linear eddy viscosity models, without ad-hoc rotation correction, in general under estimate the intensity of the bilge vortex. Best performance is seen for various anisotropic turbulence models, such as explicit algebraic stress models (EASM) [4], not yet available in STAR-CCM+, and Reynolds stress models (RSM). The Elliptic-Blending RSM [4] and the k-ω SST model [6] with curvature correction (k-ω SST-CC) [7] are applied within this study.

First, the overall wave pattern, for CFD results with k-ω SST-CC, is compared to test. For all grid refinements, and for both y+ > 30 and y+ < 1, the results are similar and in good accordance with test data; see Figure 2 for the fine grid, y+ < 1, wave profile along the hull and a wave cut at $y/L_{PP} = -0.1043$. The wave elevations are very small, but the grid surface resolution and HRIC scheme capture the wave profile, except a slightly underpredicted wave elevation close to stern. Applying RSM, a converged solution has not yet been obtained, large disturbances of the water surface are observed already upstream the hull. Instead, to be able to compare the different turbulence models, a model with the free water surface replaced by a symmetry plane (denoted double model) and the motion of the hull neglected (the sinkage and trim motions of the hull are very low) is constructed.

Comparing the full model including free surface and the double model, for both cases applying the finest grid and y+ < 1, the flow field at the stern are very similar. The total resistance is 0.47 N higher for the free surface model, which could be seen as the wave resistance and added resistance due to different sinkage and trim; this resistance discrepancy is only 1.4 % of the total resistance for the free surface model. This confirms the possibility of studying the relative differences between the turbulence models by applying a double model assumption for the JBC test case.

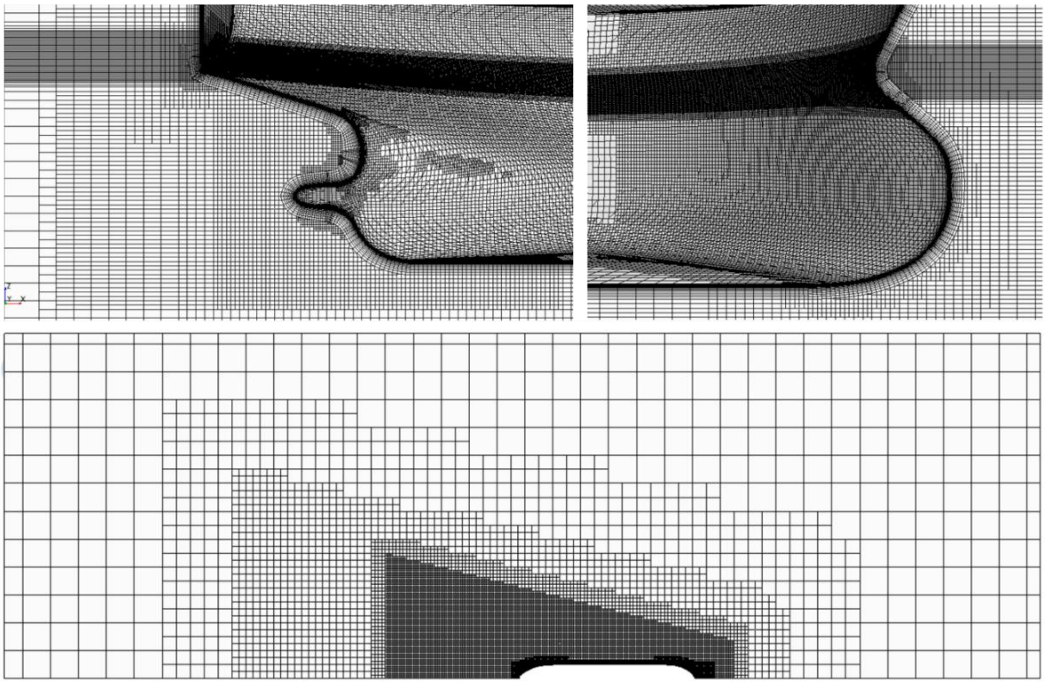

Figure 1: The structure of the coarse mesh with resolved boundary layers around the JBC hull, showing refinements near the hull, for the water surface and for the wake

The axial velocity field prediction at the stern, for the free surface model using k-ω SST-CC and the fine grid, with y+ < 1 and y+ > 30 respectively, is compared to test data for a plane located at $x/L_{pp} = 0.9625$ in Figure 3. It is clear that the wall function model fails to accurately predict the hook shape of the bilge vortex. Different prism layer configurations, fulfilling y+ > 30, have been investigated but without success. The simulations on the y+ < 1 grid predict the hook shape of the bilge vortex more accurately and is also in better accordance with other test data available further downstream.

The axial velocity, on the same plane as shown in Figure 3, is shown for k-ω SST-CC, RSM and k-ω SST, from the double model results in Figure 4. The flow field is relatively similar for k-ω SST-CC and the RSM model while the standard k-ω SST model predict a less intense bilge vortex, in a similar manner as seen for several contributions applying a linear eddy viscosity model in the 2010 workshop on ship hydrodynamics [3].

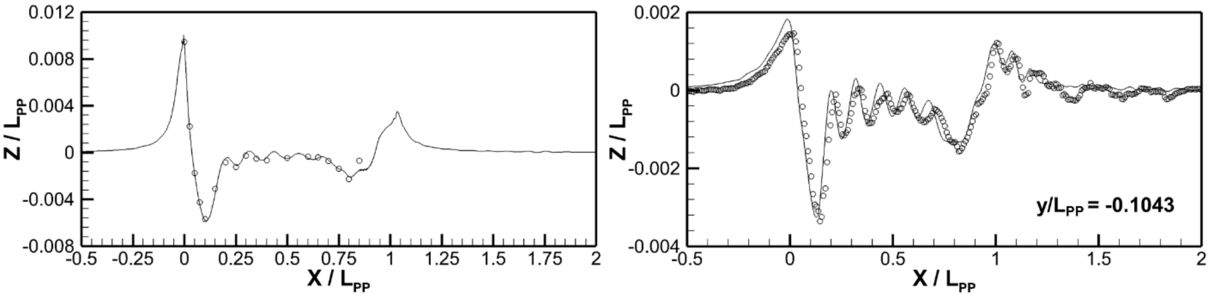

Figure 2: Wave profile along hull (left) and wave cut at $y/L_{PP} = -0.1043$ (right) for the fine grid, y+ < 1. CFD results (line) compared to test data (circles).

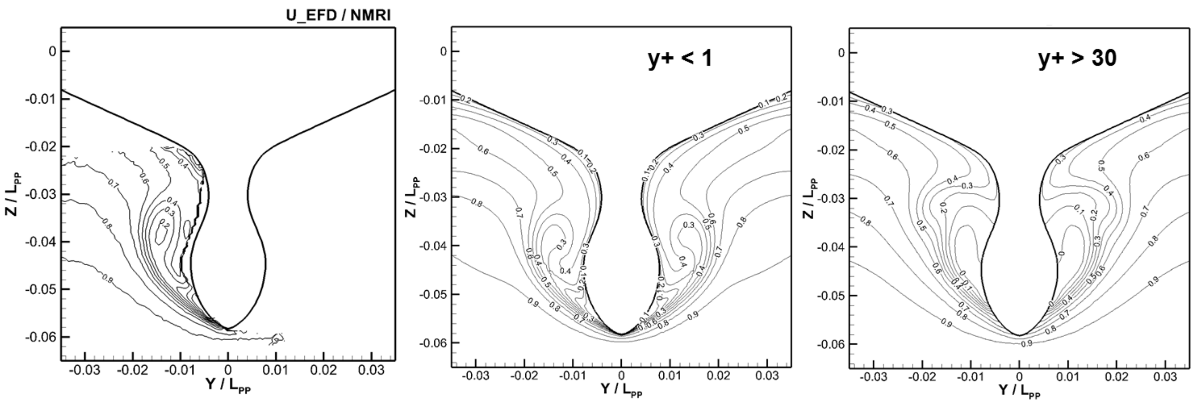

Figure 3: Mean axial velocity at $x/L_{pp} = 0.9625$. CFD results (k-ω SST-CC, fine grid, y+ < 1) compared to test.

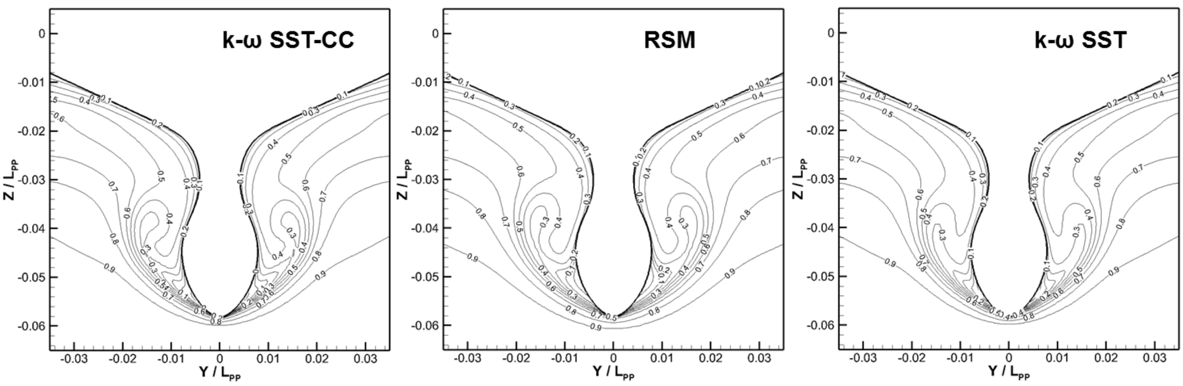

Figure 4: Mean axial velocity at $x/L_{pp} = 0.9625$. Double model CFD results (fine grid, y+ < 1) for different turbulence models.

At last, the hull resistance, sinkage and trim predictions for the free surface models, are compared to test in Table 1. The total resistance is clearly underpredicted by all models using k-ω SST-CC, while RSM predict the resistance more accurately. Note however that the RSM resistance prediction is based on a double model and that ~1.4 % is assumed to be added due to wave resistance and motion. The difference in resistance between the turbulence models can mainly be deduced from an increased wall shear stress, evenly distributed over the hull surface. Interesting to note is that the resistance prediction is in better accordance with test data for the wall function grids, with an average deviation of ~2-3 %. It has further been observed that the use of curvature correction has a minor influence on the resistance. Different sensitivity studies are conducted for the boundary layer mesh; it could be observed that a reduction of the height of the near wall cell so that $y+_{max} < 0.2$ implies a ~3 % increased resistance.

For the free surface model resistance prediction, no clear mesh convergence can be observed, but a slightly larger difference is seen between the coarse and medium grids than between the medium and fine grids. Neither for sinkage and trim can a clear mesh convergence be observed as the grids are refined. The average difference to test data is 4.2% and 4.0% for sinkage and trim respectively, compared to the mean deviation for the KVLCC2 test case from the 2010 workshop [3] of -33.3% and 7.4% respectively, these figures are in a clearly better agreement with measurements.

Table 1: Resistance coefficients, sinkage and trim compared to test. Relative differences included.

	Test	*Coarse, y+<1 VOF*	*Medium, y+<1 VOF*	*Fine, y+<1 VOF*	*Fine, y+<1 Sym.*	*Fine, y+<1 Sym. RSM*
$C_T \cdot 10^3$	4.289	3.987/*-7.0%*	3.967/*-7.5%*	3.963/*-7.6%*	3.908/*-8.9%*	4.289/*0.0%*
$C_F \cdot 10^3$	N/A	2.910	2.848	2.901	2.910	3.244
$C_P \cdot 10^3$	N/A	1.077	1.119	1.062	0.998	1.045
Sinkage [% L_{PP}], upward positive	-0.086	-0.090/*4.3%*	-0.088/*1.8%*	-0.092/*6.6%*	N/A	N/A
Trim [% L_{PP}], bow up positive	-0.180	-0.186/*3.6%*	-0.188/*4.6%*	-0.187/*3.7%*	N/A	N/A

These last results imply that currently there is no established method that both provides an accurate prediction of the stern wake as well as the resistance except from the Reynolds stress model in STAR-CCM+. Applying RSM is time consuming, an estimation based on the current case is ~2.5 times the k-ω SST simulation time, it requires a very high quality mesh and, as described above, is not straight-forward to use together with VOF.

2. Propeller open water simulation

In the open water test the inflow to the propeller is symmetrical which implies that it is possible to perform steady state simulations with only one propeller blade and utilize periodic interfaces. The grid created for one blade can then be multiplied to form a full propeller. The propeller domain is embedded in a larger stationary domain in the open water simulations, similarly to the setup for self-propulsion simulations. Mixing plane interfaces are applied between the regions.

The provided open water characteristics covers J =0.1-0.8 (Δ=0.05) and these operating conditions are simulated. The propeller rotation rate is set to 17 rps, while the advance velocity is varied to meet requested advance ratio, implying that the Reynolds number during test ($4 \cdot 10^5$) is met with -2.5% to +3.6%. The advance velocity is specified on the inlet boundary (5D_P upstream propeller) and a static pressure is set on the outlet boundary (6D_P downstream propeller). The far field outer boundary (7D_P in diameter) is modelled as a symmetry plane. The inlet turbulence intensity is set to 1 % and $\mu_T/\mu = 10$, but sensitivity studies showed minor influence on the results of these boundary conditions.

The computational grid for the propeller domain consists of polyhedral cells with prism layers along wall surfaces. Polyhedral cells are used instead of the trimmed cells due to the complex geometry with high curvature. The resulting mesh structure is shown in Figure 5, a close up of the trailing edge is also shown. A sensitivity study to the leading and trailing edge resolution was conducted, but reducing the cell surface size to 50 % and 25 % respectively, did not significantly modify the overall results (the efficiency increased with 0.1 %-points at J=0.6). The general mesh resolution was also studied and considered sufficient (a reduction of the average cell size with 50 %, implied a difference in efficiency of <0.01 %-points). The stationary domain consists of trimmed polyhedral cells. Both wall function grids (y+ > 30) and grids resolving the boundary layers (y+ < 1) have been investigated, for these 6 respectively 15 prism layers with a near wall cell height of 0.4 mm resp. 0.01 mm are applied. The wall function grid has $1.3 \cdot 10^6$ cells per propeller blade whereas the y+ < 1 grid has $2.1 \cdot 10^6$ cells.

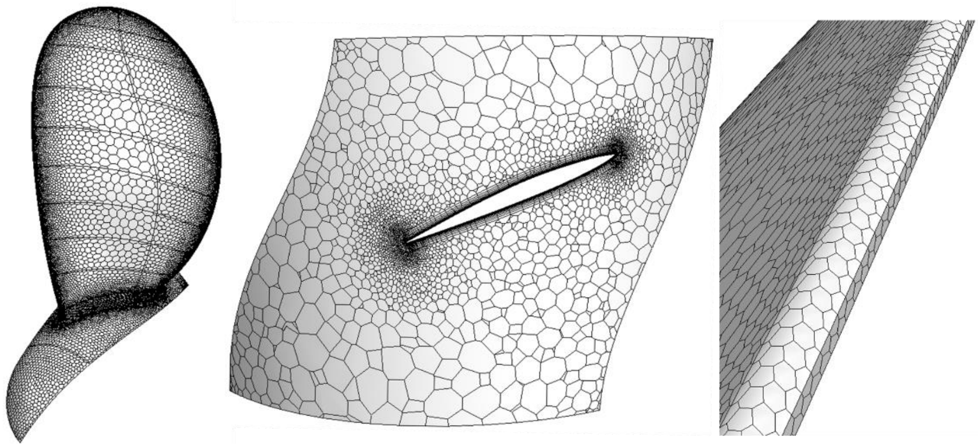

Figure 5: Structure of the y+ < 1 mesh. Surface grid, a section at 0.7R a close up of the trailing edge shown.

The CFD results, for the y+ > 30 and y+ < 1 grids, applying the k-ω SST turbulence model are compared to open water test data in Figure 6. Here the k-ω SST-CC model results are not presented, as it did not influence the overall results significantly (for instance, the efficiency increased with 0.2 %-points at J=0.6). Both CFD models are underpredicting the measured efficiency. For the y+<1 grid, which is most close to test data, this is a result of an underpredicted thrust and a less underpredicted torque. At J=0.6 the calculated thrust, torque and efficiency differs with -5.2 %, -0.7 % and -4.4 % compared to measured values, respectively.

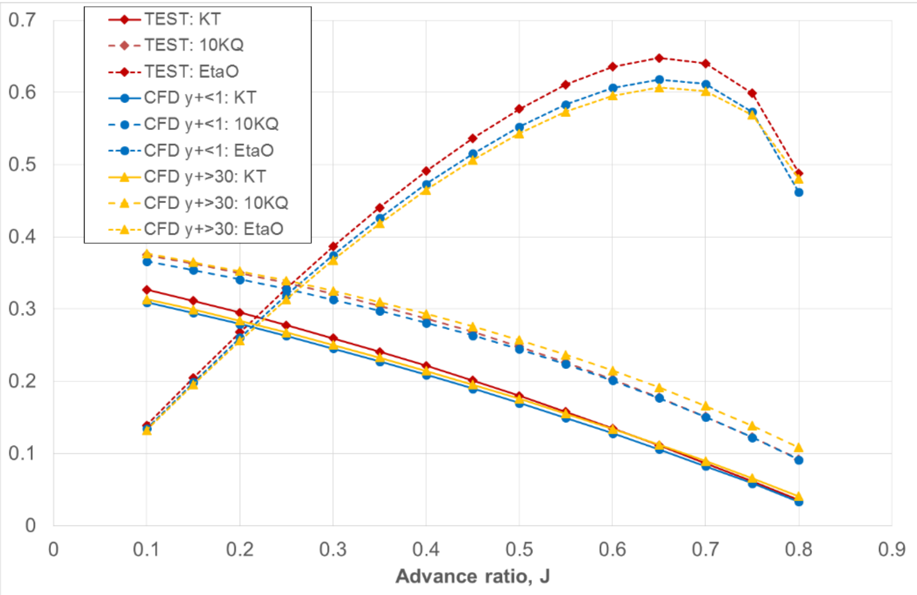

Figure 6: K_T, K_Q and η_O from open water test and CFD results.

Summing up, the differences between the simulation results and open water test data are considered to be slightly higher than expected. Since detailed knowledge about the test setup is not available, it has not been considered meaningful to further investigate the reason behind the differences. However, one possible reason is that the CFD model wrongly assumes that the boundary layer is fully turbulent (which then not was the case during the open water tests), but this is not fully clarified.

3. Self-propulsion simulation

The CFD method for the self-propulsion simulations, with a full hull and actual propeller, including a free water surface and heave and pitch motion of the hull, is tested applying the coarse grids with wall functions. The rotating motion of the propeller is simulated through the use of sliding interfaces. The time step is chosen to correspond to ~1° of propeller rotation (to speed up convergence the simulation is started with a stationary propeller using a moving reference frame and a larger time step).

The simulation, despite its complexity, is converging fine, but there are still some main issues to be addressed. Firstly, as concluded above, $y+ < 1$ are necessary to resolve the flow properly at the stern, this implies a grid size of $\sim 25 \cdot 10^6$ cells, which considering the physics solved for, is not very suitable as an engineering work tool. Simplifications, as for instance application of a symmetry plane as water surface could be considered for this case, but is not a general solution. Secondly, the method to automatically adjust the rotational speed of the propeller to match the applied rope force from test need to be developed. This is important for a proper modelling of the interaction effects.

4. Concluding remarks

For our further studies with focus on the interaction effects between propulsor and hull, there are still issues concerning the CFD methodology to be sorted out. The k-ω SST turbulence model incl. curvature correction shows promising results, but clearly underestimate the hull resistance, this could for instance be studied further applying another test case. Another strategy is to apply RSM, which showed very good accordance with measurements, it however requires more effort to get converged results for a free surface model. Applying the knowledge gained from the bare hull and open water simulations for the self-propelled hull implies a large total grid size, to facilitate further analyses possible simplifications need to be further investigated.

5. Acknowledgements

This research is supported by Rolls-Royce Marine through the University Technology Centre in Computational Hydrodynamics hosted by the Department of Shipping and Marine Technology at Chalmers. The simulations were performed on resources at Chalmers Centre for Computational Science and Engineering (C3SE) provided by the Swedish National Infrastructure for Computing (SNIC).

6. References

[1] Muzaferija, S. and Peric, M. (1999) Computation of free surface flows using interface-tracking and interface-capturing methods. In *Nonlinear Water Wave Interaction*, Marenholtz, O. and Markiewicz, M. (eds.). Southampton: WIT Press

[2] Krasilnikov, V. (2014) *Numerical Modelling of Ship-Propeller Interaction under Self-Propulsion Condition* STAR Global Conference 17-19 March 2014, Vienna

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

[4] Deng, G., Queutey, P. and Visonneau, M. (2005) Three-dimensional flow computation with Reynolds stress and algebraic stress models. In *Engineering turbulence modelling and experiments*, Rodi, W and Mulas, M. (eds.) Vol 6, pp 389-398

[5] Manceau, R. and Hanjalic, K. (2002) *Elliptic blending model: A new near-wall Reynolds stress turbulence closure* Physics of Fluids Vol 14, pp 744

[6] Menter, F.R. (1994) *Two-equation eddy-viscosity turbulence modeling for engineering applications* AIAA Journal 32(8), pp 1598-1605

[7] Arolla, S and Durbin, P (2012) *Modeling rotation and curvature effects within scalar eddy viscosity model framework* International Journal of Heat and Fluid Flow Vol 39, pp 78-89