



HAL
open science

VERIFICATION AND VALIDATION OF RESISTANCE AND PROPULSION COMPUTATION

Ganbo Deng, Alban Leroyer, Emmanuel Guilmineau, P. Queutey, Michel
Visonneau, J. Wackers, Alvaro del Toro Llorens

► **To cite this version:**

Ganbo Deng, Alban Leroyer, Emmanuel Guilmineau, P. Queutey, Michel Visonneau, et al.. VERIFICATION AND VALIDATION OF RESISTANCE AND PROPULSION COMPUTATION. Numerical Ship Hydrodynamics, 2015, Tokyo, Japan. hal-02572184

HAL Id: hal-02572184

<https://hal.science/hal-02572184>

Submitted on 13 May 2020

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

VERIFICATION AND VALIDATION OF RESISTANCE AND PROPULSION COMPUTATION

G. Deng, A. Leroyer, E. Guilmineau, P. Queutey, M. Visonneau & J. Wackers (ECN-LHEEA, CNRS, France)
A. del Toro Llorens (Spanish Institution of Civil Engineers, Spain)

1. SUMMARY

This paper is devoted to the verification and validation exercises with the ISIS-CFD code conducted for the Tokyo 2015 workshop. In addition to numerical uncertainty estimation, we also address issues such as the effect of turbulence modelization, wall resolved versus wall modeled approach, propeller resolved versus propeller modeled approach for self-propulsion simulation, etc.

2. INTRODUCTION

CFD can be considered as a mature tool now for steady state ship hydrodynamic applications such as resistance in calm water. Accurate enough predictions can be obtained with reasonable resources even for fully appended hulls, both for model and for full scale in a routine design procedure. However, rigorous V&V exercises are seldom performed by CFD users. Most of cases, one grid and one computation are adopted following guidelines based on recommendations and user experiences. The recommended setup such as grid density, turbulence model, etc can be different from one institution to another. Comparison with measurement data is often the only criterion when establishing those guidelines. The versatility of a guideline thus established can be questionable, since a small comparison error can be the result of error cancellation between numerical discretization and physical modelization error. By performing a careful V&V exercise, we attempt to quantify turbulence modelization error and try to answer to questions such as whether a non-linear turbulence model is more accurate than a linear turbulence model for ship resistance prediction, what is the impact on the accuracy when wall function is used.

Compared with resistance computation, validation for propulsion computation is much more challenging. To author's knowledge, the only approach that is capable to predict ship propulsion with accuracy is to simulate directly the rotating propeller with sliding grid or overset approach. Time accurate simulation is required for such simulation even if time averaged solution is sufficient. Our experiences during the V&V exercises for this workshop show that a reliable numerical uncertainty estimation is nearly impossible for such simulation due to high iterative error as well as time discretization error. Self propulsion simulation can also be performed by modeling the effect of propeller by body forces in the RANSE solver. With such an approach, propeller thrust can be provided by the RANSE solver. But to determine propeller revolution rate and propeller torque, a simplified model or a

coupling approach between RANSE solver and another specific solver simulating the propeller such as RANSE/BEM coupling approach must be used. In the present paper, a simplified model is employed.

3. NUMERICAL APPROACH AND CASE SETUP

Computation has been performed with the ISIS-CFD flow solver developed by our team, also available in the commercial software FINETM/Marine. It is an unstructured finite volume RANSE solver using free-surface capturing approach. Due to the limited length of this paper, the technical details of the solver will not be described here.

Except for the case when propeller motion is resolved by the RANSE solver, only half domain is simulated. The inlet boundary is located at 2.5Lpp from FP, while the outlet is located at 3.0Lpp after AP. Bottom and top boundaries are located at 1.5Lpp and 0.5Lpp from the waterline, respectively. Lateral boundary is located at 1.5Lpp from the mid plane. Pressure boundary condition is applied at the bottom and top boundaries, while far-field boundary condition is applied at the inlet, outlet, as well as the lateral boundary. We rely on Richardson extrapolation for the V&V exercise. It is well known that Richardson extrapolation can be applied only when grid similarity is ensured. The unstructured hexahedral mesh generator HexpressTM available in FINETM/Marine is employed in the present study. With HexpressTM, it is not possible to generate a set of rigorously similar grids. But with a special setup, it is possible to ensure grid similarity before the insertion of viscous layer. Our experiences show that grid thus generated can usually make possible a successful Richardson extrapolation. This grid generation setup is too specific to the grid generator HexpressTM and will not be described here. Interested readers can refer to the master thesis by A. del Toro (2015) who has performed most of the computations presented in this paper during his master thesis in ECN. Table 1 gives the number of grid cells for different grid sets used in the present study for the JBC test case.

Table 1 Number of grid cells for different cases

Cases	Grid4	Grid3	Grid2	Grid1
1.1a_wm	405K	1.512M	3.143M	5.724M
1.1a_wr	861K	2.632M	5.304M	9.197M
1.2a_wm	725K	2.311M	4.806M	8.750M

1.2a_wr	1.317M	4.269M	8.344M	14.077M
1.5a_wm	2.442M	4.784M	10.247M	18.676M
1.6a_wm	2.513M	6.668M	13.913M	25.332M

In Table 1, "wm" stands for wall modeled simulation for which wall function approach is used, while "wr" stands for wall resolved simulation for which low Reynolds turbulence model is used. For the first case, the same y^+ value about 30 is applied for all grids, while for the second case, y^+ value changes from about 0.4 for the coarsest grid to about 0.16 to the finest grid. Meshes for different configurations have similar grid density. The difference in number of cells is due to the presence of energy saving device (ESD) and the propeller, additional cells in the viscous layer when using wall resolved approach, and whole domain simulation rather than half domain simulation. Mesh density is not too fine. Mesh size near the free-surface is about $0.0008L_{pp}$ for the fine mesh. Grid 1 and grid 2 represent a mesh commonly used for resistance computation for engineering application. Unless otherwise stated, all computations have been performed with the non-linear EASM turbulence model. Second order upwind blended scheme is employed for spatial discretization except for the case with propeller resolved simulation for which a more stable ALVSMART scheme is used.

4. RESULTS AND DISCUSSIONS

4.1 Resistance Results for the JBC test cases

Detailed V&V results can be found in the proceedings of the workshop. Main results for total resistance are given in Tables 2 and 3 for case 1.1a (without ESD) and 1.2a (with ESD) respectively. In this table, we give only the finest grid solution U1, the observed order of convergence p , Richardson extrapolation error RE% defined as $(\delta_{RE}-U1)/\delta_{RE} \cdot 100$, and the comparison error E%D defined as $(D-S)/D \cdot 100$ where D is the measurement data, $S=U1$ is the simulation result, δ_{RE} is the result of Richardson extrapolation. The least squared approach proposed by Hoekstra & Eca (2008) is used for Richardson extrapolation. When the observed order of convergence is higher than 2.1, Richardson extrapolation is obtained with assumed second order accuracy. For both cases, the EASM model gives better prediction than the SST model. Moreover, numerical discretization error is smaller than the difference due to turbulence model for the fine grid. Hence, when the grid is fine enough, the EASM model should give better prediction for ship resistance for this test case. The reason for the better performance with the EASM model is due to the existence of a relatively strong aft-body vortex for this geometry. When aft-body vortex is not so strong, the SST model should also be capable to give an accurate prediction for ship resistance as well. We can also notice that even with a fine grid containing more than 6M cells, numerical discretization error for resistance computation is still about 2% at least. Hence, when the grid is further refined, the EASM model is expected to under-estimate the resistance by about 4% for the case without ESD, and 3% for the case with ESD. This is confirmed by computations with adaptive grid refinement which give a comparison error of 3.1% for the case without ESD, and 2.2% for the case with ESD. For both cases, the use of wall function does not deteriorate too much the predicted result. The predicted resistance differs only by 0.1% and 0.45% respectively, which is much smaller than the discretization error. This observation justifies the use of wall function for engineering application due to much lower computation cost. Flow separation

is observed on the ESD. This possibly explains why the comparison error, the Richardson extrapolation error, and the observed order of convergence are higher for the case 1.2a when wall function is used.

Table 2 Total resistance for case 1.1a

Simulation	U1	p	RE%	E%D
easm_wm	4.209	2.07	-2.3	1.87
easm_wr	4.213	1.94	-2.0	1.77
sst_wr	4.087	1.59	-3.2	4.71

Table 3 Total resistance for case 1.2a

Simulation	U1	p	RE%	E%D
easm_wm	4.200	2.93	-4.3	1.48
easm_wr	4.219	2.06	-2.3	1.03
sst_wr	4.093	1.67	-3.2	3.99

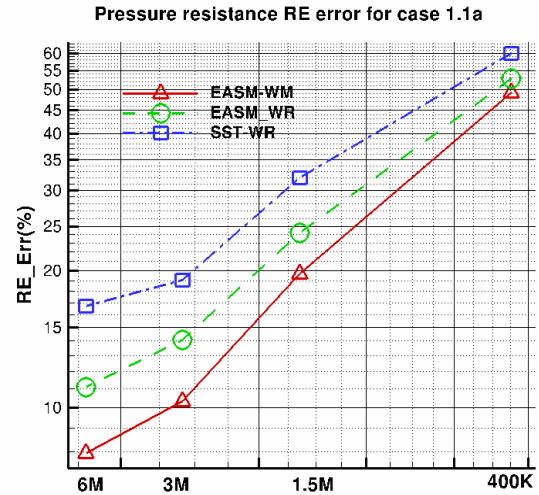


Fig.1 Richardson extrapolation error for pressure resistance.

Predicting pressure resistance with good accuracy is a challenging task for CFD. Figure 1 displays Richardson extrapolation error for pressure resistance for the case without ESD. Even with the finest grid, the error is still about 10% for the EASM model. Much higher uncertainty is observed for the SST model. But such high level of numerical uncertainty might due to low observed order of convergence (1.53). As pressure resistance represents only about 25% of the total resistance, numerical error observed in total resistance comes mostly from pressure resistance error. For applications in which the contribution of pressure resistance becomes more important, such as vessels with smaller L/B ratio, higher grid resolution might be needed to achieve acceptable accuracy.

4.2 Self-Propulsion Results for JBC test cases

The most obvious approach to perform a self-propulsion computation is to simulate the rotating propeller with the RANSE solver using sliding grid or overset grid approaches. A sliding grid approach is employed in our computation. With such approach, time accurate simulation is required even when only time averaged results is needed. A rigorous V&V study with such an

approach requires numerical uncertainty estimation on space and on time. Due to high computational cost, no attempt is made to assess time discretization error during the present study. Instead, time step as well as non-linear iteration number per time step are chosen according to open water computations using the same grid for the propeller. Sliding grid approach gives almost the same result for propeller thrust compared with a computation performed in rotating frame. This "calibration" gives 150 time steps per revolution and 15 non-linear iterations per time step. We perform a first computation with a large time step to accelerate the ship to the target speed until convergence. Rotating frame approach is applied to the propeller domain. Ship trim and sinkage are computed during this computation. Then, in a restart computation, we switch to a small time step (150 time steps per revolution). Ship motion is frozen during this computation and this is why, during this restart, ship dynamic position is not computed accurately.

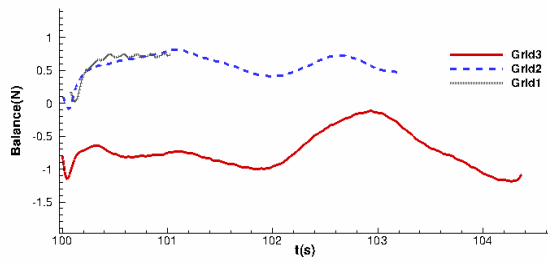


Fig.2 Force imbalance for case 1.5a (without ESD)

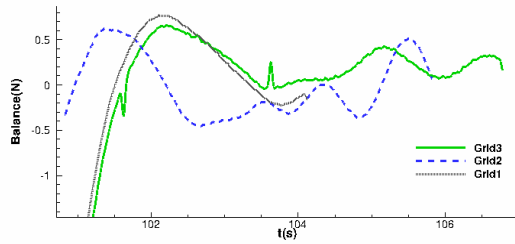


Fig.3 Force imbalance for case 1.6a (with ESD)

In our propeller resolved simulation, computations have been performed with the EASM model using wall function only. Computations have been performed on 4 grids with different grid density as the cases for resistance computation. Figures 2 and 3 show the evolution of force imbalance in our simulation for case 1.5a and 1.6a respectively. 0.5N imbalance represents about 1.2% ship resistance. Force imbalance is expected to be zero under self propulsion condition. The raw data are highly fluctuating due to rotating propeller. Results shown in the figure are smoothed data by applying 1000 passes with the smoothing operation available in Tecplot post-processor. Force imbalance obtained on the coarsest mesh are not shown in the figure. It is very high (about 8N). Such high level of force imbalance is due to the fact that very strong flow separation occurs at stern, resulting in a highly asymmetric wake. In our simulation, propeller revolution rate is prescribed with the measurement value. Propeller thrust is positive. For the case without ESD, force imbalance has a positive sign on the fine mesh (Grid1), which means that propeller thrust is too high. We need to reduce propeller revolution rate to satisfy the self propulsion condition. For the case with ESD, we are not too far from the self propulsion condition. For case 1.6a, we have performed about 7 seconds physical time, namely more than 50 propeller revolutions.

With 150 time steps per revolution and 15 non-linear iterations per time step, the CPU cost is equivalent to about 30 resistance computations. Yet, it is hardly possible to determine the converged value for force imbalance. Due to such a convergence behavior, we believe that iterative error in our simulation is much too high compared with discretization error. Hence, it is impossible to perform any reliable uncertainty estimation for discretization error.

Table 4 Comparison error for propeller resolved simulation

	Case 1.5a		Case 1.6a	
	Value	E%D	Value	E%D
Ct*1000	4.661	3.11	4.572	3.99
Kt	0.214	1.47	0.227	2.78
Kq	0.029	-5.55	0.031	-3.52

Table 4 presents the predicted results with the finest grid for Ct, Kt and Kq as well as relative error compared with measurement data. In spite of high numerical uncertainty, predicted results are reasonable. High propeller torque is a typical result for RANSE simulation when turbulence transition is not simulated. But as we can see in the following section that accuracy of wake flow prediction can be the cause of such over prediction as well. It should be noticed that propeller thrust and ship resistance are not clearly defined in a propeller resolved RANSE simulation. They are evaluated during post-processing using a procedure that is not always clearly defined. Concerning our result, we consider the dynamic axial force acting on the propeller domain as propeller thrust. This choice is justified by the fact that propeller thrust thus obtained agrees with the simulation using actuator disk approach presented later in this paper. With this post-processing procedure, we underestimate propeller thrust and ship resistance compared with measurement data. If we consider axial force acting on propeller blades as propeller thrust, then for case 1.6a, we will over predict propeller thrust by 1.2% and underestimate ship resistance by 2%. This results in a better agreement with measurement data, while it is exactly the same simulation result.

We have also performed self propulsion simulation by using body force approach with actuator disk model. Propeller thrust can be determined directly from the RANSE computation. But to determine other quantities related to propeller performance such as propeller torque and propeller revolution rate, a special coupling procedure is required. The RANSE solver can be coupled with BEM code or other type of simplified code to simulate the action of the propeller. In the present study, we employ a simpler approach without using any other simplified code. We only use the open water Kt-Kq result obtained from the measurement to determine the missing quantities with post-processing. The procedure is as follows. First, we perform a usual RANSE computation with actuator disk approach to simulate the effect of the propeller. Propeller thrust is adjusted during this computation such that self-propulsion condition is satisfied. After having obtained the converged solution with the RANSE solver, we compute the total velocity at the propeller plane. The total velocity is computed on a disk with the same size as the propeller diameter. This will give us two conditions: propeller thrust and total velocity. We will perform an additional open water computation using actuator disk approach based on the open water Kt-Kq result. In this open water actuator disk computation, propeller revolution rate and propeller advancing speed is adjusted such that propeller thrust

determined from the K_t - K_q result and the total velocity computed at the propeller plane are the same as the values obtained with the RANSE computation with the hull. With two conditions and two unknowns, the problem is well defined and can be easily solved iteratively. Compared with more complex coupling procedure such RANSE/BEM coupling approach, there is no need to compute the propeller induced velocity.

Table 5 Propeller modeled simulation for case 1.5a

	Wall resolved		Wall modeled	
	Value	E%D	Value	E%D
$C_t \cdot 1000$	4.625	3.87	4.620	3.97
K_t	0.214	1.24	0.213	1.84
K_q	0.0291	-4.41	0.0291	-4.19
n (tps)	7.60	2.56	7.62	2.31

Table 6 Propeller modeled simulation for case 1.6a

	Wall resolved		Wall modeled	
	Value	E%D	Value	E%D
$C_t \cdot 1000$	4.660	2.14	4.617	3.04
K_t	0.2385	-2.36	0.2327	0.13
K_q	0.0306	-3.66	0.0305	-3.25
n (tps)	7.31	2.53	7.33	2.27

Unlike the case for resistance computation, we are unable to obtain a result with good convergence behavior with respect to the requirement for Richardson extrapolation. For this reason, only the predicted C_t , K_t , K_q and propeller revolution rate n obtained with the finest grid as well as the relative error compared with measurement data are shown in table 5 and table 6 for the cases without and with ESD respectively both for wall resolved simulation and for wall modeled simulation using wall function. Unlike the case with propeller resolved simulation, propeller thrust and ship resistance are clearly defined in the propeller modeled RANSE computation. Compared with measurement data, predicted results are slightly better than what we obtained with much expensive propeller resolved simulation presented in table 4. As the computation is performed with half domain, propeller tangential force is not taken into account. Error due to this approximation needs to be investigated in a future study. In our simulation, experimental K_t - K_q result is employed to determine propeller torque coefficient K_q and propeller revolution rate n . It can be seen that propeller torque is over-predicted as the case with propeller resolved simulation. In spite of the uncertainty about the accuracy of such simplified approach, we believe that such over prediction of propeller thrust can be attributed to the accuracy of the predicted wake. As we can see in the following sub-section, predicted axial velocity at propeller plane is smaller than the measurement result, especially for the case without ESD. This explains why estimated propeller revolution rate is lower, and propeller torque is higher. In both cases, wall resolved simulation and wall modeled simulation give about the same accuracy. This result justifies once again the use of wall function for engineering applications.

4.3 Wake Flow Results for JBC test cases

The mesh set employed in the present study is designed to ensure an accurate enough accuracy for ship resistance and propulsion prediction based on our experiences. Spatial resolution in the wake near the propeller plane is about $0.00086L_{pp}$ with the finest grid. It turns out that with such grid resolution, the difference of the predicted axial velocity contours obtained with the two finest grid is still clearly visible as shown in figure 4.

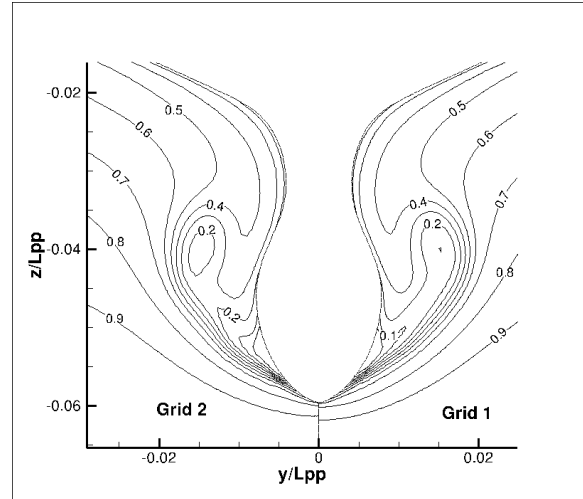


Fig.4 Predicted U velocity contours at section S2.

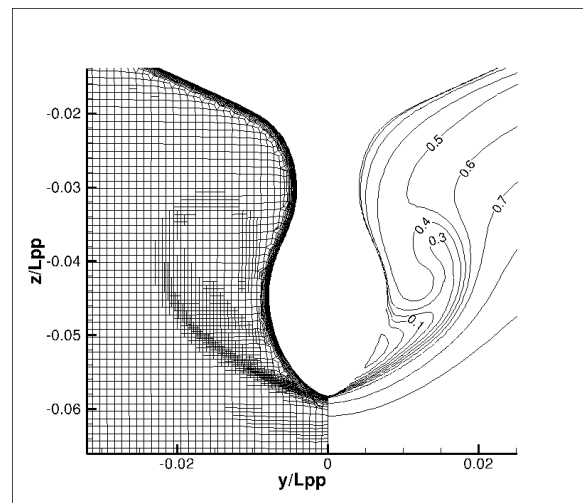


Fig.5 U velocity contours obtained with double model

We attempt to obtain a more accurate solution with adaptive grid refinement. Results obtained with double model computation using wall resolved EASM are shown in figure 5. The adaptive mesh contains about 35M cells. Comparison with measurement data is shown in figure 6. In the core of aft-body vortex, the predicted axial velocity is higher than the measurement value, while for free-surface computation, the predicted value is lower. This is the reason why we over-predict propeller torque in the computation with free-surface. The higher predicted axial velocity in the core of the vortex might be due to double model computation. We have performed another adaptive grid refinement computation with free-surface. The minimum cell size

is refined to about 0.00009Lpp. We discover that with such a fine grid, flow instability develops. Due to such unexpected unsteadiness, predicted wake flow is quite different from what we obtained when the numerical solution converges to a steady solution. Such unsteadiness is also observed when the mesh is refined manually in the wake with similar grid resolution. But the amplitude of unsteady fluctuation is not exactly the same. Due to the uncertainty in reproducing such unsteadiness, we prefer not to submit the wake flow result with grid refinement to the workshop. But the predicted free-surface elevation with adaptive grid refinement is submitted.

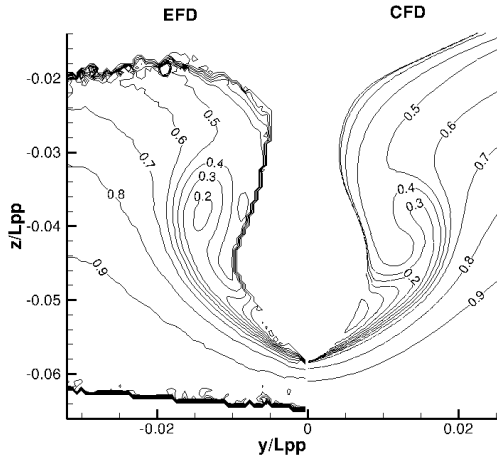


Fig.6 Comparison of U velocity contours at section S2

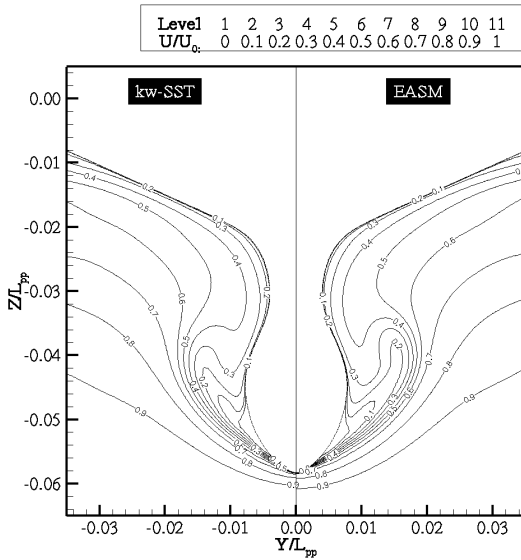


Fig.7 Comparison between SST and EASM model

It is interesting to observe that with a coarser grid (grid 2), the agreement between the CFD prediction and the measurement result is better. Better agreement is also obtained for resistance. With grid2, we have 1% under-estimation, while with grid 1, the error is about 2%. We believe that it is the effect of error cancellation. On the coarse grid, numerical discretization error cancels with turbulence modelization error. We have not performed any propulsion computation with the SST model in the

present study. But by checking the mean axial velocity at the propeller plane, we can observe that the predicted mean axial velocity using the SST model is about 5% higher compared with the EASM model. Hence, if a propulsion simulation is performed with the SST model, then error in propeller torque should be smaller, and the propeller revolution rate will be higher. But we can not conclude that the SST model is more accurate for propulsion computation. By comparing the predicted axial velocity contours at section S2 between the SST model and the EASM model, we can observe that the SST model fails to predict the location for the core of the aft-body vortex correctly.

4.4 Simulation for the ONR Tumblehome Test Cases

V&V study was not performed for the ONR Tumblehome test case. It is a very challenging 6-DOF free run CFD computation with active rudder control in waves. To determine the propeller revolution speed, a calm water self-propulsion computation at model point needs to be computed first. For this configuration, both propeller resolved and propeller modeled computations have been performed. The propeller resolved simulation is performed with half domain using sliding grid. The mesh contains 9.3M cells with 865K cells in the propeller domain. The propeller modeled simulation is performed with a whole domain using overset grid to handle rudder motion. The background grid with the ship contains 6.5M cells. Each overset grid for the rudder contains about 750K cells. Measurement data are available for trim, sinkage and propeller revolution rate only. Comparison with measurement data is shown in table 7. All computations have been performed with the SST model.

Table 7 Calm water results for ONR Tumblehome

	Sinkage*100m		Trim (deg)		n (rps)	
	Value	E%D	Value	E%D	Value	E%D
Prop-R	0.234	-3.3%	-0.044	-13.3%	8.73	2.6%
Prop-M	0.245	-8.3%	-0.034	11.0%	8.83	1.5%

Error observed for trim angle is unusual. Propeller resolved and propeller modeled approach give quite different results and both CFD predictions are far from the measurement data. We believe that such discrepancy is mainly due to the approximation made in our computation. For the propeller resolved simulation, ship free-motion is computed only in the first computation using ship motion adapted time step with rotating frame approach applied for the propeller domain. In the second restart computation with rotating grid approach applied for the propeller domain, ship motion is frozen. Predicted result shows that the vertical force acting on the propeller is quite different between the two computations. Moreover, the vertical force acting on the propeller on the trim angle is important. Trim angle shown in table 7 is a corrected result based on the predicted vertical force for the case with propeller resolved simulation. In the propeller modeled simulation with actuator disk, propeller vertical force is not taken into account, which results in an under estimation of trim angle. The predicted resistance coefficient, K_t and K_q are $5.092e-3$, 0.249 , 0.0753 and $5.041e-3$, 0.232 , 0.0674 for propeller resolved simulation and propeller modeled simulation respectively. Measurement data is not available for those quantities. But we believe that the propeller torque is over estimated with a propeller resolved approach.

Free run computations have been performed with propeller modeled approach for head sea and 45 deg bow sea only. The following setup procedure is applied. First, a trim and sinkage free only captive motion with prescribed approach speed is performed until a well established solution is obtained. Then, when wave crest arrives at bow, the ship is released and a free run computation is performed with rudder angle controlled by the autopilot law prescribed in the instruction. Propeller revolution speed is kept unchanged with the value determined from the calm water self-propulsion computation presented above. Unfortunately, we have made a mistake in moment of inertia in our setup. X moment of inertia is 52% lower, while Y and Z moment of inertia is 24% higher. This will have an important impact on our predicted results, especially for ship speed and roll motion. As each computation takes more than one week, we did not have time to repeat the computation with a correct setup.

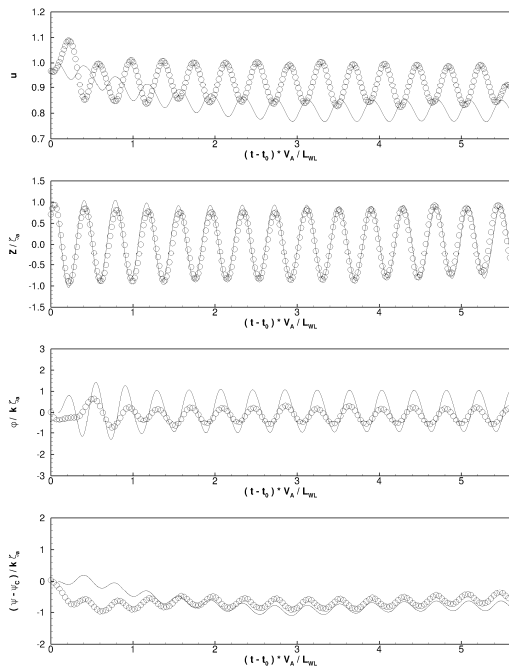


Fig.8 Ship speed and motions for bow sea configuration

Figure 8 presents the predicted ship speed as well as heave, roll and yaw motions for the bow sea configuration. Measurement data are shown by symbols. Lower ship speed and higher roll motion amplitude are certainly the consequence of wrong moment of inertia in our setup. A correct computation will be performed and the result will be presented during the workshop. In our computation, the course of the ship is not maintained at the expected trajectory. It is not surprising. We believe that the prescribed autopilot law is not capable of maintaining the course. Rudder force predicted with propeller modeled approach employed in our simulation must not be accurate enough. This explains why we can not maintain the course with the same control law used in the measurement.

5. CONCLUSION

The V&V exercise for the JBC test case confirms what we have learned from previous benchmark test case such as HSVA and KVLCC2 for U-shape vessel. Turbulence model is unable to simulate aft-body vortex with a perfect agreement. Consequently, ship resistance is slightly under-predicted. According to our estimation, this under estimation is about 7-8% and 3-4% for a linear and non-linear turbulence model respectively for the case without ESD, which is however acceptable. However, it has been noticed many times that the error obtained with a mesh containing a few million cells in our CFD computation is usually less than 2% when using the EASM model. This is probably due to error cancellation. Inspection of pressure resistance reveals that numerical error is still about 10% for this quantity with grid resolution commonly used for industrial application. For a different hull form with smaller L/B ratio, higher grid resolution is therefore required. Physical modelization error is also expected to be higher. For self-propulsion computation, we were unable to obtain a reliable estimation for numerical uncertainty both with propeller resolved and with propeller modeled approach. However, comparison error is always in the range of a few percents as for the resistance. Hence, we consider that the quality of numerical prediction is also reliable. Although there is no doubt the non-linear EASM model can provide better prediction for ship resistance and wake flow compared with the linear eddy-viscosity SST model, it is not sure that this model also performs better for self propulsion computation. Better prediction of the EASM model can be observed on axial velocity distribution as well as on the location and intensity of the aft-body vortex. However, we feel that propeller prediction is more sensitive to mean axial velocity in the propeller plane rather than to the detailed velocity distribution, at least when propeller modeled approach is employed. Further investigation is required concerning this issue. In the present study, we also found that a very simple propeller modeled approach can give equally accurate prediction for ship propulsion compared with the expensive propeller resolved approach. Error due to the use of wall function is very small compared with numerical discretization error and turbulence modelization error. It can be used for similar engineering application with confidence. With a mesh containing less than 10M cells, we have found that it is hardly possible to obtain a grid independent solution for wake flow. Attempt to obtain better accuracy with adaptive and manual grid refinement leads to the development of flow unsteadiness in the case of JBC, while such behavior is not observed in the computation for the KVLCC2 test case. We hope to clarify this issue in a future study.

Acknowledgement: This work was granted access to the HPC resources under the allocation 2015-2a1308 made by GENCI (Grand Equipement National de Calcul Intensif)

REFERENCES

- del Toro Llorens A. (2015), "CFD Verification and Validation for Ship Hydrodynamics", *Master thesis, Ecole Centrale de Nantes*
- Hoekstra M, Eca L. (2008), "Testing Uncertainty Estimation and Validation Procedures in the Flow Around a Backward Facing Step", *3rd Workshop on CFD Uncertainty Analysis*, Lisbon.