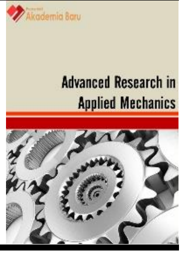
Journal of Advanced Research in Applied Mechanics 58, Issue 1 (2019) 1-10

Penerbit Akademia Baru

Journal of Advanced Research in Applied Mechanics

Journal homepage: www.akademiabaru.com/aram.html
ISSN: 2289-7895



An Investigation into Numerical Uncertainty of 17.500 DWT Model Tanker Resistance

Open Access

Dian Purnamasari[1,*], I Ketut Aria Pria Utama[1], I Ketut Suastika[1]

[1] Departement of Naval Architecture, Institut Teknologi Sepuluh Nopember Surabaya, Indonesia

ABSTRACT

The credibility of computational simulations is related to the quantifying the error. The convergence study is a reduction of the numerical errors associated with simulation parameters. The numerical simulation of 17.500 DWT tanker resistance in calm water conditions at Fr= 0.13 - 0.20 has been investigated by ITTC standard. Two different turbulence models EASM and k-ω SST were implemented and deviations in the numerical results are highlighted. The verification is performed by assessing grid quantity and Richardson extrapolations, and uncertainty analysis with the factor of safety methods. The numerical results of total resistance coefficient of two turbulence model were compared with the experimental data. These numerical investigation shows well captured free surface, the wave elevation around the hull are in qualitative agreement with the experimental.

Keywords:
resistance, verification, Richardson extrapolations, factor of safety, uncertainty analysis

1. Introduction

Over the past few years, computational fluid dynamics (CFD) has been constantly developing new numerical methods. There has been a growing interest in CFD techniques applied to ship hydrodynamics for both commercial and research codes. Stern *et al*., [6] reviewed the progress made over the last 10 years in CFD applied to ship hydrodynamics. In some countries the CFD simulations have commonly been integrated into project of every new vessel, especially in the design of seagoing ships. Larsson *et al*., [5] gathered the results of 33 groups and 18 cases in the Gothenburg 2010 Workshop on Numerical Hydrodynamics. The credibility of computational simulations is related to the quantifying the error, meanwhile understanding of the sources of the uncertainty can provide guidance on how to reduce or manage uncertainty in the simulation. The convergence study is a reduction of the numerical errors associated with simulation parameters. Yao *et al*., [12] discusses the definition, the sources, the classification and the expressions of the CFD uncertainty.

During the last fifteen years, much progress has been made in the development of robust and accurate computational strategies enabling simulation of the water flow around ships. Free surface phenomenon around a ship hull plays an important role in its resistance. Almost all methods used

* *Corresponding author.*
E-mail address: dian15@mhs.na.its.ac.id (Dian Purnamasari)

are based on the Navier-Stokes equations. The discretization methods for the water surface differ widely. Queutey and Visonneau [9] built a new scheme for Blended Interface Capturing Scheme refined under the acronym for Blended Reconstructed Interface Capturing Scheme to keep constant the width of the interface on the smallest number of control volumes by reducing the numerical diffusion and dispersion, to ensure a monotonic change of the volume fraction. The ship resistance considering free surface presented by Pranzitelli *et al*., [8] was employed to simulated the free surface flow around a semi-displacing motor yacht hull. Shia *et al*., [11] plenty of numerical simulations for the resistance of ship model under the different running of the draft-trim coupled form and Froude numbers are carried out.

The current study aims to establish the resistance a numerical technique to simulate flow around the model hull of tanker 17.500 DWT at Froude number of 0.13 to 0.20. This work describes to calculate resistance force at steady forward with free surface simulation using Volume-Of-Fluid (VOF) formulation. Multi-phase analyses have been done in this study by using the interface capturing approach that solves the RANSE equations on a predetermined grid which covers the entire domain. The multi-block grids and the effect of two turbulence model EASM and k-ω-SST on the results are being used to obtain numerical results.

2. Model Geometry

The geometry studied is a model scale of the tanker 17.500 DWT. The main particulars are given in Table 1. This hull has bulbous bow in the fore and stern in the aft, also a large block coefficient (C_B). The hull geometry is depicted in Figs. 1-2.

Table 1
Main particulars of the tanker 17.500 DWT

Particulars	Ship	Model
Length between perpendiculars	149.500 m	5.980 mm
Breadth	27.700 m	1.108 mm
Draft	7.000 m	0.280 mm
Displacement volume	23.464 m^3	1.502 m^3
Wetted surface area	5.307 m^2	8.491 m^2
Block Coefficient	0.809	0.809
Scale Ratio		1/25

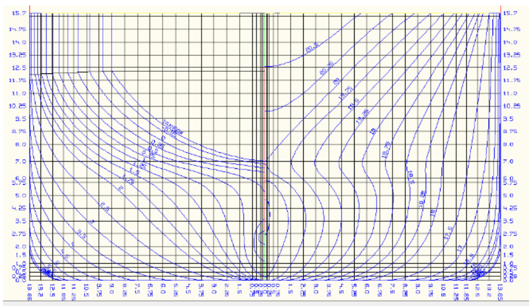


Fig. 1. Body Plan of the tanker 17.500 DWT

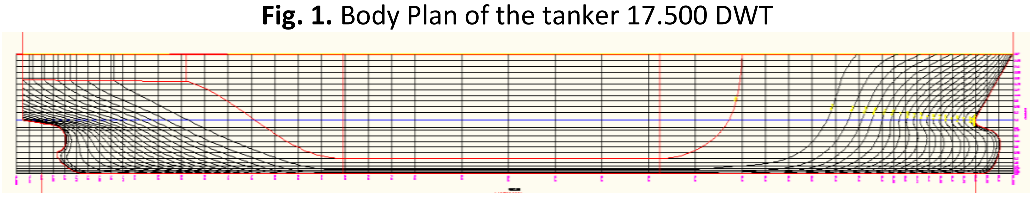


Fig. 2. Profile of the tanker 17.500 DWT

3. Numerical Setup

The total resistance of a model scale hull was computed using CFD simulations based on the incompressible RANS equations. The simulations were performed with the FINE/Marine flow solver also including the unstructured Cartesian mesh using finite volume method formulation mesh generator HEXPRESS™, which is aimed at the simulation of realistic flow problems in all branches of marine hydrodynamics (Numeca, 2013).

A computational domain was created in the rectangular shape around the model and boundary condition shown graphically in Figure 3. The dimensions from the model in terms of L_{pp}: inlet 1xLpp, outlet 3xLpp, Top 1xLpp, Bottom 2xLpp, Each side1.5xLpp. The fluid flows along x-direction and positive y-axis points the starboard direction while z-axis points upward. A far field boundary condition was applied at side, inlet and outlet boundaries. A slip wall condition was applied to the deck. A prescribed pressure boundary condition was applied to the top and bottom boundary.

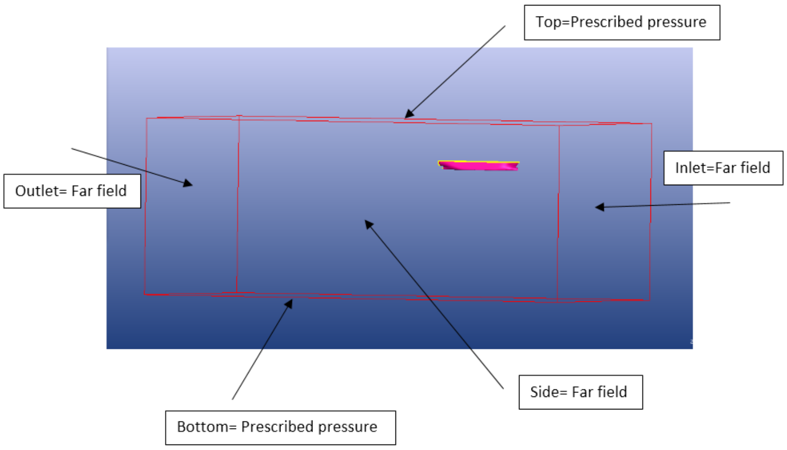


Fig. 3. Representative domain boundaries

The grid is generated in five steps: initial mesh, adaptation to the geometry, snap to geometry, optimize mesh and viscous layer addition. Mesh generation to create unstructured hexahedral grids, an initial coarse grid which in the following is refined at the boundaries by subdividing initial cells. Then a mesh optimization is started to improve the quality of the cells. Finally, viscous layers are inserted on designated boundaries, see Numeca [7]. Figure 4 show refinement around the geometry features and around the free surface, resulting in three different meshes, with a cell count ranging from 1.4 million to 3.9 million

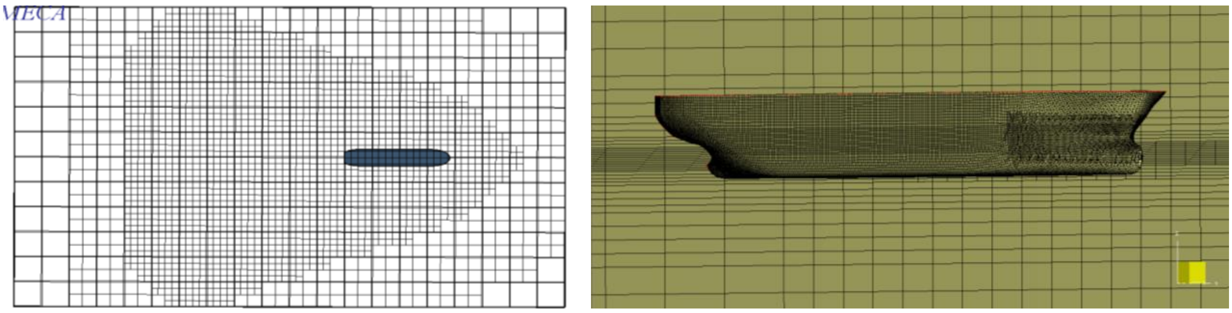


Fig. 4. Generated unstructured full-hexahedral mesh

k-ω SST and EASM turbulence models are used with standard wall functions to model the Reynolds stress term. The ITTC guidelines [3] recommend placing the first grid point at a distance from the ship's wall such that 30 < y+ < 100.

The computations settings common to time scheme: backward order, multi-fluid computation: water: dynamic viscosity: 8.51 *(Pa.s)x10^{-4}*, density: 996.5 *kg/m^3* and air: dynamic viscosity: 1.85 *(Pa.s)x10^{-5}*, density:1.2 *kg/m^3*. Simulation control variables as follows: 500 timesteps, uniform time step = 50 sub-cycles, 8 non-linear iterations. The simulation was setup as a steady state solution, fixed in trim and heave according to the conditions of the experimental data.

With the increased demands in the complication of simulated models and the accuracy of their results, sophisticated methods are needed to assess the quality of the computational fluid dynamics

(CFD) simulations. Convergence studies for grid spacing was undertaken to assess numerical errors for three solutions are used with refined mesh on the fine, medium, and coarse grids and uncertainties with grid sizes developed by Roache [10]. This method is currently used and recommended by ASME. Convergence studies of 3 solutions to evaluate convergence with respect to input parameter, the convergence ratio RG is defined as

$$R_G = \varepsilon_{21}/\varepsilon_{32} \tag{1}$$

where $\varepsilon_{21} = S_2 - S_1$ and $\varepsilon_{32} = S_3 - S_2$ give the change of solutions between the medium-fine and coarse-medium grids.

Using generalized Richardson extrapolation, the first-order RE estimate δ^*_{RE} and the order of accuracy P_G can be estimated as follows

$$\delta^*_{RE_{G,1}} = \frac{\varepsilon_{21}}{r_G^{p_G} - 1} \tag{2}$$

$$p_G = \frac{ln(\varepsilon_{32}/\varepsilon_{21})}{ln(r_G)} \tag{3}$$

A factor of safety approach used to define the uncertainty where an error estimate from RE is multiplied by a factor of safety to bound simulation error followed the ITTC recommendation

$$U_G = F_S|\delta^*_{RE_{G1}}| \tag{4}$$

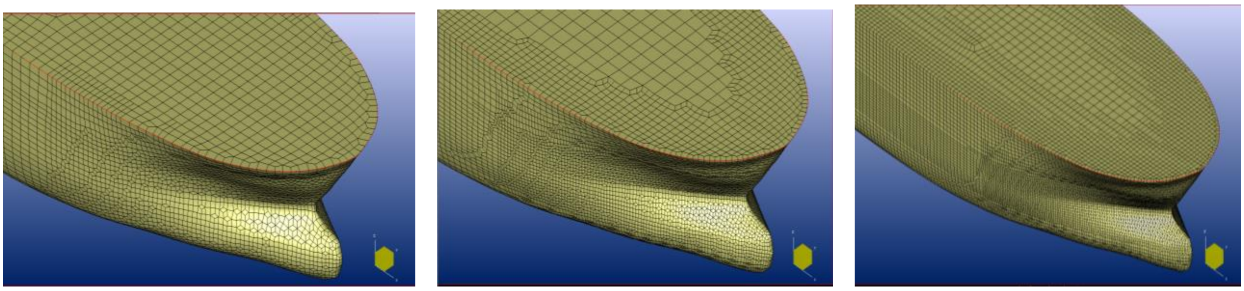
4. Results

Table 2 shows results of grid convergence analysis study based on changing the initial cell sizes. Three volumic refinement boxes are added to the initial mesh. Different initial meshes will lead to different mesh sizes. This allows to refine the whole fluid volume and not only the areas right next to the solid.

Table 2
Number of cells for the mesh dependency study

	Initial mesh			Number of Cells
	x	y	z	
Coarse	10	6	4	1.487.007
Medium	15	9	6	2.513.477
Fine	20	12	8	3.942.362

Figure 5 (a.b.c) show the complete generated mesh for coarse, medium and fine mesh respectively. A wall function implementation is used in boundary conditions of the ship, viscous prismatic cells around the hull consisted of 20 layers with an expansion ratio of 1.2.



(a) (b) (c)

Fig. 5. The complete generated mesh

A comparison is conducted between the most widely used turbulence models k-ω SST and EASM. The percentage errors of two turbulence models are shown in Figure 6. The error value of the total resistance coefficient from SST model is less than 3% for fine grid, while the error value from EASM model more than 7%. This shows that a simple turbulence model may give a better prediction for model ship resistance compared with more advanced turbulence model. EASM turbulence model effect on resistance prediction varies.

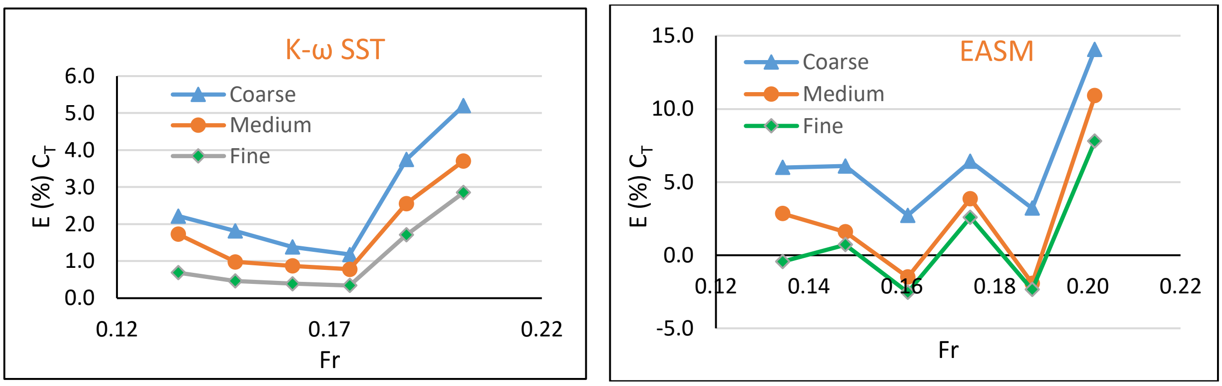


Fig. 6. Percentage comparison errors for the two different turbulence models

The wave elevation along the hull has been measured and plotted, along a section situated at y/Lpp=0.148. Results are compared the calculation with the EASM and kω-SST models. The charts below compare wave elevation along x obtained with different turbulence model. Both models give qualitatively the same results.

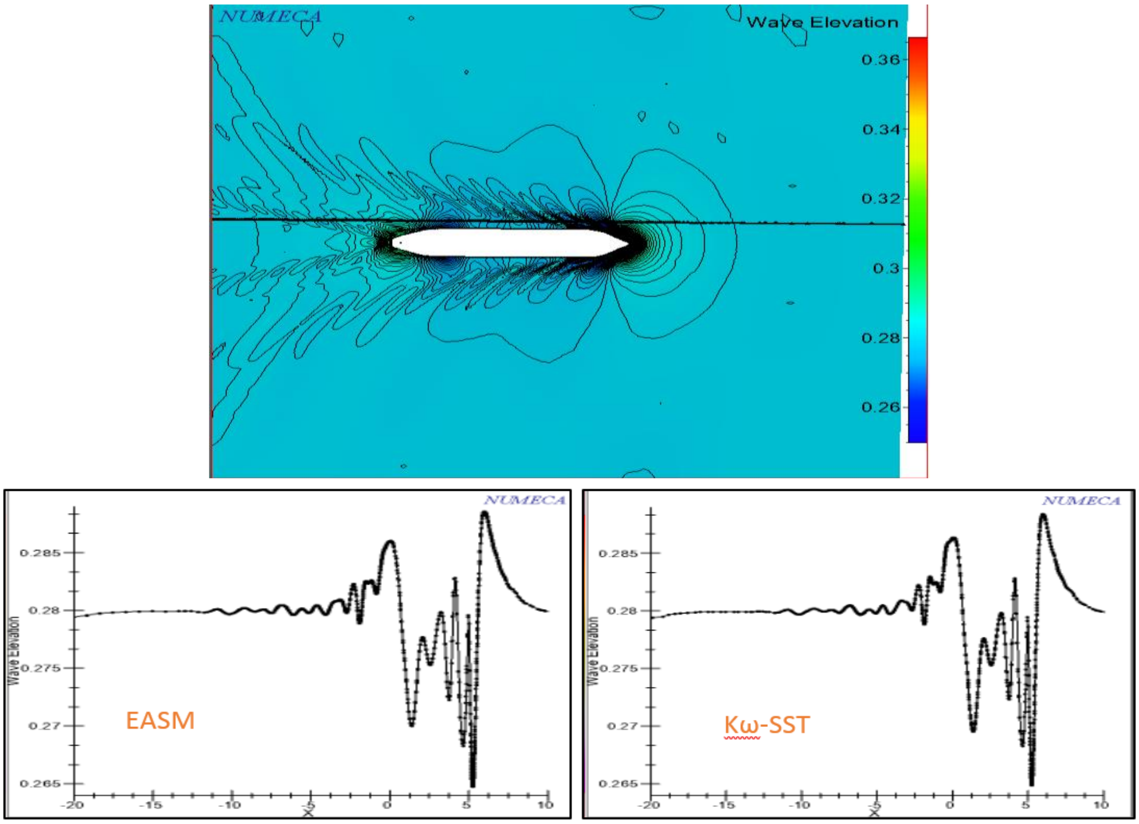


Fig. 7. Wave elevation at y/Lpp=0.148

The result of the isolines and local value were obtained from two turbulence model EASM and K-ω-SST, the free surfaces are match perfectly and quite the same.

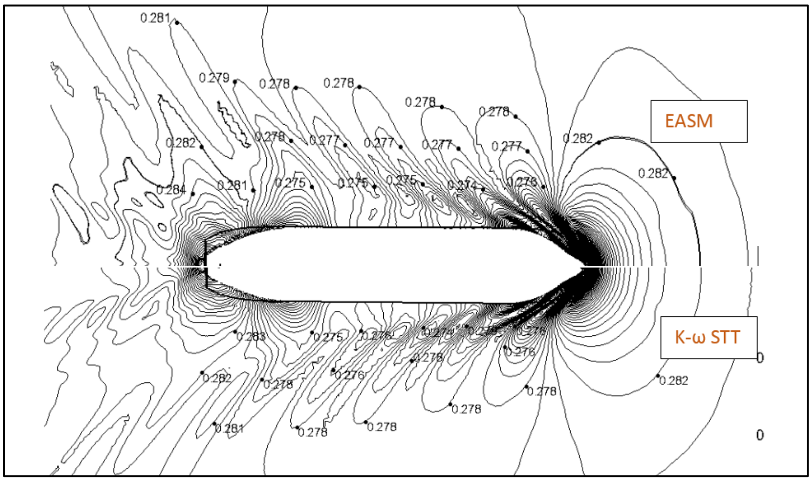


Fig. 8. The isolines and local value

The standard turbulence models of two-equation, eddy-viscosity models and explicit algebraic Reynolds-stress models have a little effect on the wetted surface as shown in Figure 9. As indicated the predictions with advanced turbulence models did not show appreciable improvements over those obtained using two-equation, see also ITTC [4].

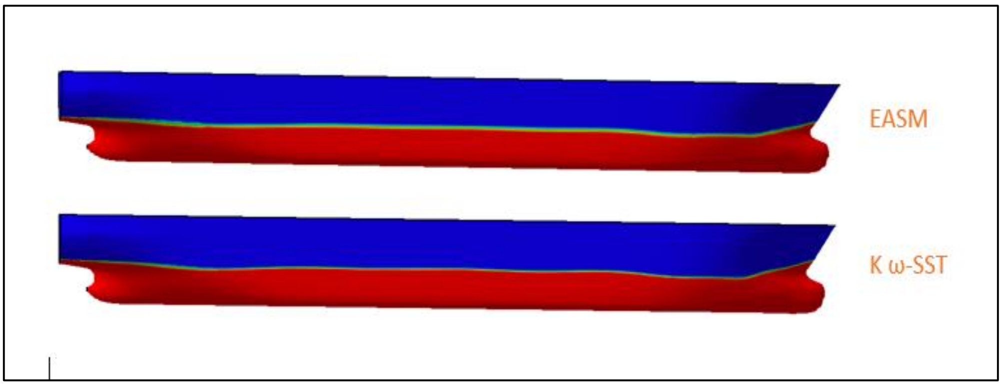


Fig. 9. Wetted surface

The contour plots of hydrodynamic pressure acting on the hull surface qualitatively colored with the velocity magnitude. The pressure shows large values near the bow region due to the effect of hull geometry. The value in the mid-ship is nerly zero such that the resistance force oscillations are small for most of the body except near the bow as shown Figure 10.

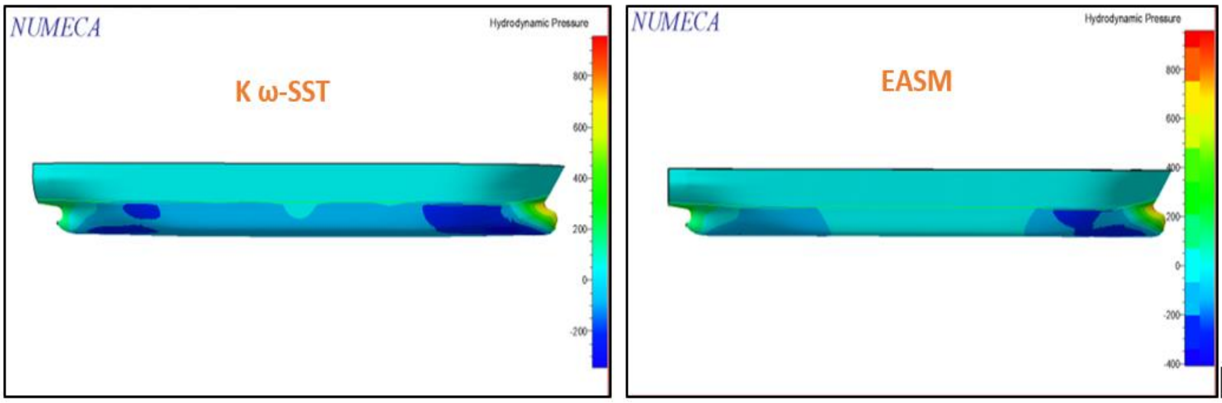


Fig. 10. Hydrodynamic pressure

Figure 11 show a graphical representation of the y+ variations on the ship model for *Fr*=0.17. The value of the first cell next to the wall should be in the range of 30-100 for the wall function types of meshes as recommended ITTC [4]. And it is seen that the *y+* values is ranged between 0<y+<30, it can be concluded that the values of y+ are within acceptable limits.

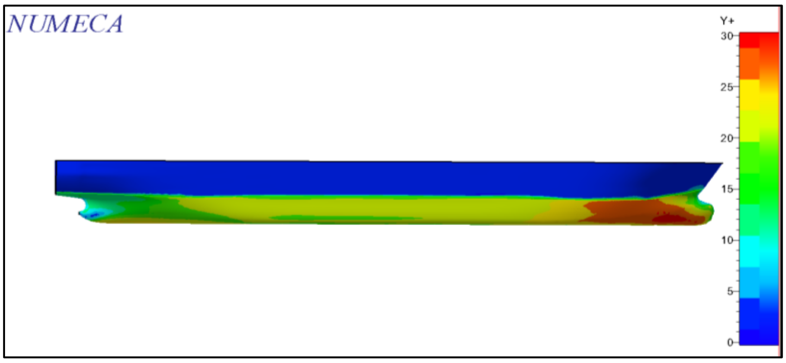


Fig. 11. Wall Y+ distribution on the hull surface (b)

Figure 12 shows result comparison for total resistance coefficient C_T. For the highest value of *Fr*, total resistance coefficient has larger differences value.

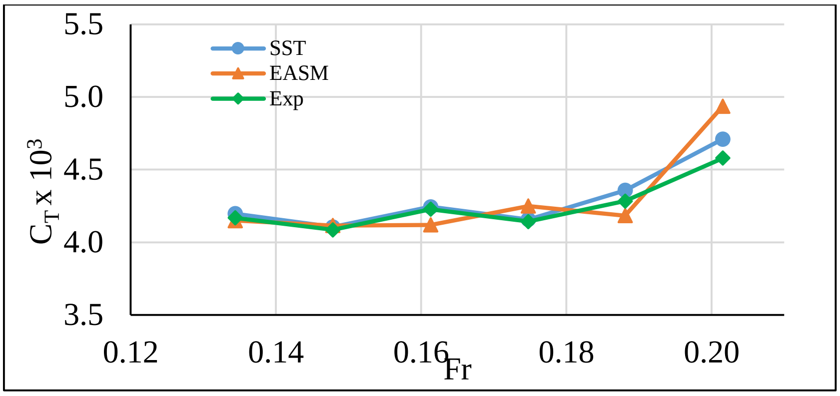


Fig. 12. Total resistance coefficient

The results of turbulence model analysis show that the turbulence model Kω-SST gives a better solution and a well agreement with the experimental data. Relative deviation for fine mesh between the EFD data and CFD simulation are lower than 3.0% for the total resistance coefficient as shown Table 3. The greatest relative deviation is 2.86% and the smallest deviation is 0.34% for Kω-SST.

Table 3
Relative deviation for fine mesh between the EFD data and CFD

Fr	K ω-SST	EASM	Exp	(Relative deviation, %)	
				K ω-SST	EASM
0.13	4.20	4.15	4.17	0.69	-0.43
0.15	4.11	4.12	4.09	0.46	0.72
0.16	4.24	4.12	4.23	0.39	-2.53
0.17	4.16	4.25	4.14	0.34	2.59
0.19	4.36	4.18	4.28	1.71	-2.34
0.20	4.71	4.94	4.58	2.86	7.81

The solutions for C_T indicate monotone convergence has been achieved with $R_G < 1$ for the design Froude number Fr = 0.17. The observed order of accuracy $P_G = 2.039$ is closer to the theoretical value ($P_G = 2$). The numerical uncertainty (U_G) less than 2%. Resistance result for turbulence model Kω-SST obtained with the fine grid is accurate enough.

Table 4
Verification of the total resistance *CT* for Kω-SST

R_G	P_G	δ^*_{RE}	U_G
0.496	2.039	1.25%	1.56%

A comparison between the experiment and steady state simulations show good agreement for the resistance. Figure 13 show wave elevation is quite good, from the analysis, it can be concluded that the numerical model and experiment are reliable and reasonable.

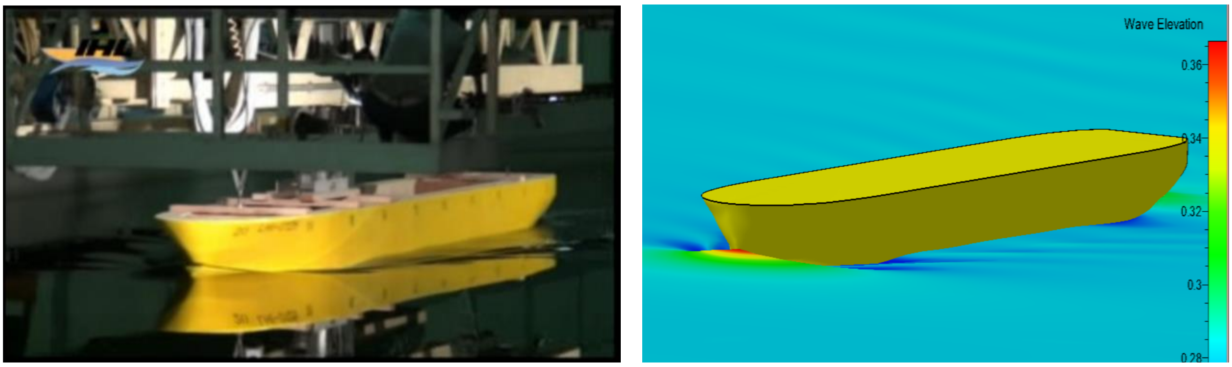


Fig. 13. Photographs between the experiment and the numerical calculations

5. Conclusion

The excellent agreement between the model tests and CFD predictions for the total ship resistance in calm water condition results in a good confidence level in the CFD results presented. By using of turbulence models and wall functions resulted in wide variation to the final prediction. Mesh fineness requires being compatible with the turbulence model/wall functions being used to obtain meaningful results. The mesh quality has proven to be an important issue in the computations.

User skill and experience are important in making proper engineering judgment based on the simulation results of such a complicated problem as the free surface flow around the ship's hulls.

Future plans focus on the comparison of these simulation results using different RANSE methods, with the results of practical measurements.

Acknowledgement

The first author, expressed her gratitude to the Ministry of Research Technology and Higher Education of the Republic of Indonesia (Ristekdikti) which funded her PhD program at ITS under contract number 07/S3/D/PTB/XI/2015

The authors would like to thank members of staff at Indonesian Hydrodynamic Laboratory r for their sincere support during the experiment and manufacture of the ship model.

References

[1] FINE™/Marine v3 Flow Integrated Environment for Marine Hydrodynamics. Documentation v3.1a, 2013
[2] Menter, Florian R. "Two-equation eddy-viscosity turbulence models for engineering applications." *AIAA journal* 32, no. 8 (1994): 1598-1605.
[3] ITTC. 2014. Practical Guidelines for Ship Resistance CFD. 7.5–03 –02–04.
[4] ITTC, 2014. Report 27th Specialist Committee on CFD in Marine Hydrodynamics.
[5] Larsson, Lars, Frederick Stern, and Michel Visonneau, eds. *Numerical ship hydrodynamics: an assessment of the Gothenburg 2010 workshop*. Springer Science & Business Media, 2013.
[6] L. Stern, S. Bhushan, T. Xing, M. Visonneau · J. Wackers · G. Deng, L. Larsson *Numerical ship hydrodynamics: an assessment of the Gothenburg 2010 workshop*. Springer Science & Business Media, 2013..
[7] Numeca (2013). User Manual Hexpress.
[8] Pranzitelli, A., C. Nicola, and S. Miranda. "Steady-state calculations of free surface flow around ship hulls and resistance predictions." *High Speed Marine Vehicles (IX HSMV), Naples, Italy* (2011).
[9] Queutey, Patrick, and Michel Visonneau. "An interface capturing method for free-surface hydrodynamic flows." *Computers & fluids* 36, no. 9 (2007): 1481-1510.
[10] Roache, Patrick J. *Verification and validation in computational science and engineering*. Vol. 895. Albuquerque, NM: Hermosa, 1998.
[11] Shia, Aiguo, Ming Wu, Bo Yang, Xiao Wang, and Zuochao Wang. "Resistance calculation and motions simulation for free surface ship based on CFD." *Procedia Engineering* 31 (2012): 68-74.

[12] Yao, Zhen-qiu, Hong-cui Shen, and Hui Gao. "A new methodology for the CFD uncertainty analysis." *Journal of Hydrodynamics* 25, no. 1 (2013): 131-147.