Journal of Ocean, Mechanical and Aerospace
-Science and Engineering-, Vol. 19

Effect of Internal Grids Structure on the Numerical Prediction of the Free Surface Flow around Wigley Hull Form

Yasser M. Ahmed[1, 3], O. B Yaakob[1,2,*], A. H. Elbatran[1, 4], Mohamed Walid Abdel-Hamed[4]

[1] *Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310, UTM Skudai, Johor, Malaysia*
[2] *Marine Technology Center, Universiti Teknologi Malaysia, 81310, UTM Skudai, Johor, Malaysia*
[3] *Dept. of Naval Architecture and Marine Engineering, Faculty of Engineering, Alexandria University, Alexandria, Egypt*
[4] *Arab Academy for Science and Technology and Maritime Transport, Abu Qir, Alexandria, Egypt*

*Corresponding author: omar@fkm.utm.my

Abstract

Two grids with different internal mesh structure have been used to predict the incompressible free surface flow around the Wigley hull form at *Fr* = 0.2 and 0.267. The finite volume RANSE code Ansys CFX, which using the two-phase Eulerian-Eulerian fluid approach has been used to perform the different numerical simulations. The Shear Stress Transport (SST) turbulence models has been used in the RANSE code. Ansys Meshing and ICEM CFD grid generators have been used to generate the two unstructured tetrahedral grids for this study. The results compare well with the available experimental data for the hull resistance at the two speeds. In addition, wave patterns, pressure contours and the time required for the numerical simulations of the grids have been compared in this study.

KEY WORDS: *free surface, Wigley hull, turbulence model*

1. Introduction

Wave making is one of the most effective resistance on the ship movement. It represents 20 to 50% of the total ship resistance depending on the ship hull geometry and ranges of Froude number. Hence, many researchers gave more attention on studying the pattern of waves generated by a ship moving through the water. Moreover, these waves contain energy that must be dissipated to the surrounding fluid.

Michell [1] and Havelock [2] studies are the oldest and important work on studying waves around ships. These research works proposed the main features of ship waves and the analytical studies on the prediction of waves around simple bodies, or hulls of ship [3]. Hess and Smith [4] and Dawson [5] used the Boundary Element Method (BEM) or panel method to investigate wave patterns around ship hulls. Jensen and Soding, [6]; Larson et al. [7] , Gatchell *et al.* [8] and Tarafder *et al.* [9] also simulated the wave resistance problems by using other technique which is Rankine source method.

The free surface flow around ship hulls can be predicted by Navier-Stokes Equations. Hino [10] predicted the free surface around ship hulls by using Navier-Stokes Equations. Dawson [5] presented double model solution which succeed to incorporate the free surface boundary condition in the simulation process. Furthermore, the interface-tracking and the interface-capturing are the most popular methods which used for RANSE simulations to investigate the free surface flow. In the interface-tracking methods, the fluid governing equations are solved in one fluid domain (water) only. The computational grids are deformed so as to fit the shape of the free surface using different types of grid-generation techniques such as the surface fitting grid method [11]. In other methods, e.g. the volume of fluid (VOF) (Hirt and Nicholas, [12]; Maronnier *et al.* [13]; Yong *et al.*, [14]), both air and water are considered and treated as two effective fluids. These methods are known as interface-capturing and deal with numerical grids fixed in space, predicting the location of the free surface by solving an additional transport equation. Hino and Takanori [15], Alessandrini and Delhommeau [16], Qiuxin and Gao [17] and Zhang *et al.* [18] carried out different numerical simulations to study free surface flow around different ship hull forms, by solving the governing equations of Reynolds Averaged Navier-Stokes equations (RANSE) code.

In this research work the finite volume RANSE code Ansys CFX has been used for simulating the incompressible free surface flow around the Wigley hull form at *Fr* = 0.2 and 0.267. Two unstructured tetrahedral grids of different internal structure have been used in this study to assess their effect of the predicted free surface flow of the ship hull. The finite volume code is using the VOF method. The SST turbulence model was used in the different simulations.

2. Mathematical Models

The mathematical model for the free surface flow in Ansys CFX is based on the homogenous multiphase Eulerian-Eulerian fluid approach. Both fluids (water and air) in this approach share the same velocity fields and other relevant fields such as temperature, turbulence, etc., and they are separated by a distinct resolvable interface. The local equations governing the motion of steady, viscous, incompressible fluid are the Navier–Stokes equations, which in a conservative formulation are given as:

$$\nabla \cdot (\rho V) = 0.0 \quad (1)$$

$$\nabla \cdot \left(\rho V \otimes V - \mu\left[\nabla V + (\nabla V)^T\right]\right) = -\nabla p + f \quad (2)$$

where,

$$\rho = \sum_{\alpha=1}^{2} r_\alpha \rho_\alpha \quad \mu = \sum_{\alpha=1}^{2} r_\alpha \mu_\alpha \;, \quad \sum_{\alpha=1}^{2} r_\alpha = 1 \quad (3)$$

and f represents the external forces. The Shear Stress Transport (SST) turbulence models were used in Ansys CFX.

3. Boundary Conditions and Computational Method.

The Wigley hull in all simulations was treated as a wall of no-slip boundary condition. The static pressure was defined as a function of water volume fraction at the outlet boundary. The coordinates system is translated on the bottom of the tank; the distance from the origin to the free surface is then defined and the static pressure computed accordingly. The initial location of the free surface was imposed by defining the volume fraction functions of water and air at the inlet and outlet boundaries. The atmospheric pressure was set as a reference pressure. Due to the symmetry of the ship hull only the starboard side of it was considered in the different simulations.

The finite volume method was used with the CFD code for the discretization process. The fluid flow was considered steady in the simulations. The advection terms have been discretized using the high resolution numerical scheme [19]. A linear interpolation scheme was used for interpolating the pressure, while the velocity was interpolated using a trilinear numerical scheme. The RMS criterion (root mean square) with a residual target value of 0.0001 was used for checking the convergence of the solutions.

4. Grid Generation and Simulations Execution Time

The free surface flow around the Wigley hull form (Fig. 1) has been predicted numerically at two different speeds (Fr of 0.2 and 0.267). The Reynolds number changed according to the change in Froude number.

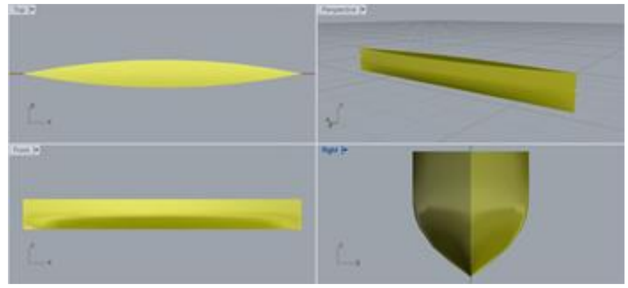

Figure 1. Different plan views for the Wigley hull.

The study focused on investigating the effect of changing the internal structure of tetrahedral grids on the prediction of hull resistance and wave pattern at different speeds. Ansys Meshing and ICEM CFD grid generators have been used in this research work to generate the required grids for the ship hull. The computational domain in both cases (Fig. 2) extends for *2L* in front of the ship hull to *3L* behind the hull, *2L* to the side and approximately *1.9L* under the still water surface. The air layer extends *0.094L* above the still water surface. The main domain was consisted of the both cases by two sub-domains. The details of mesh elements of the two grids can be seen in Table 1 and Figs. 3 and 4.

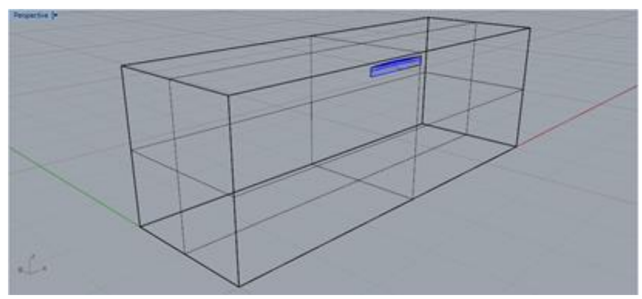

Figure 2. Study computational domain

Table 1 Properties of the grids used in this study

Grid	Hull mesh size (m)	Method of joining sub-domains
Grid1 (Ansys Meshing)	≈ 0.002	Using interface
Grid2 (ICEM CFD)	≈ 0.001	Using common surface (interior)

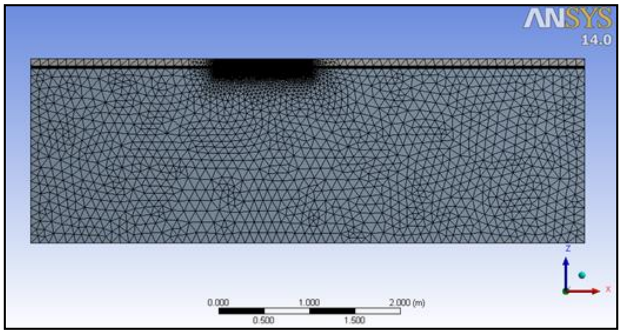

Figure 3(a). Grid1 with prism layers in the free surface region and on WIgley hull (total number of mesh elements is 1,598,573 tetrahedral elements)

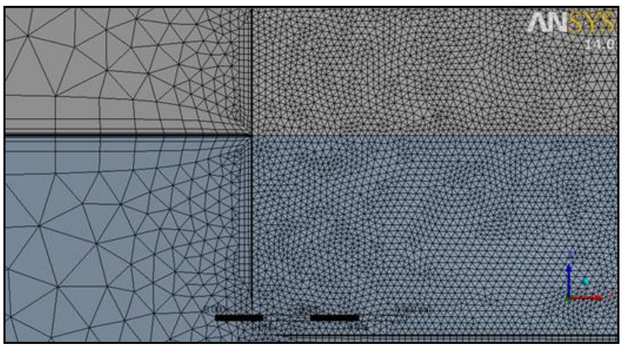

Figure 3(b). Close view for the prism layers on Wigley hull and free surface area for Grid1.

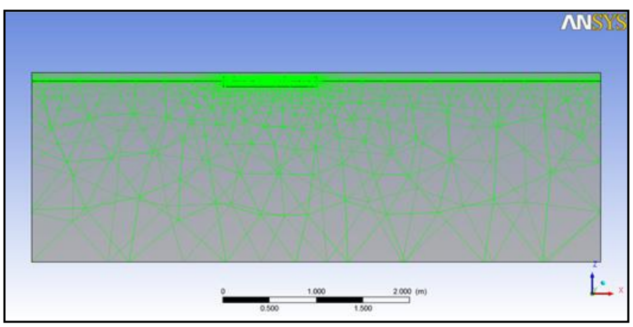

Figure 4(a). Grid2 with prism layers in the free surface region only (total number of mesh elements is 3,731,145 tetrahedral elements)

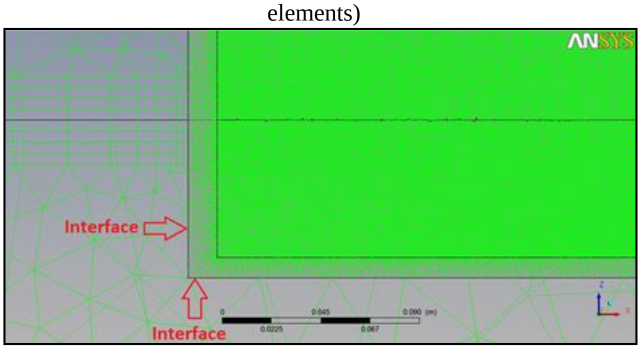

Figure 4(b). Grid2 interfaces plans between the sub-domains. No prism layers in the ship internal domain for the free surface.

The numerical results for CFX were obtained on a single computer Core i7 (8 cores) 3.4 GHz and 12 GB RAM. The execution time for Grid1 and Grid2 simulations is given in Table 2. In spite of, Grid2 has nearly double the number of mesh elements of Grid1, it needs approximately half the execution time for the other grid at the two speeds. This give indication about the stability that is provided by the internal structure of grid2 to the CFD solver during the different simulations.

Table 2 Execution time for Grid1 and Grid2

Case	**No. of elements**	***Fr***	**Execution Time (*h*)**
Grid1	1,598,573	0.2	12.72
		0.267	11.69
Grid2	3,731,145	0.2	7.41
		0.267	6.55

5. Results and Comparisons

The results obtained for the total resistance coefficient, $C_T = R_T / 0.5\rho V^2$, where R_T is the hull total resistance, ρ is fluid density and V is the ship velocity at the two speeds are given in Figs. 5 and 6. The results showed the over-prediction of Grid1 to the resistance of the hull at lower *Fr* (0.2), while Grid2 gave better prediction for the hull drag at this speed as shown in Fig. 5. However, the difference between the numerical results of Grid1 and the experimental results is 0.06 N, and 0.022 N for Grid2, which is considered in general good prediction for both grids at this lower speed. At *Fr* = 0.267 (*V* = 0.8363 m/s), Grid1 nearly gave the same value of C_T such as the experimental one (0.021 N difference), while Grid2 calculated value is relatively far from experimental results (0.192 N difference).

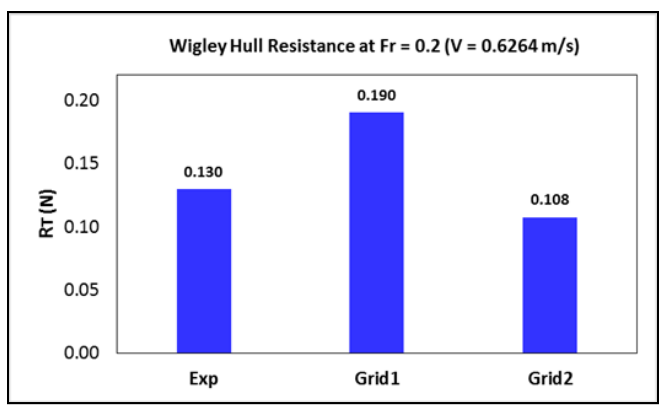

Figure 5. Comparison between Wigley hull experimental [20] and predicted numerical results at *Fr* = 0.2 (*V* = 0.6264 m/s).

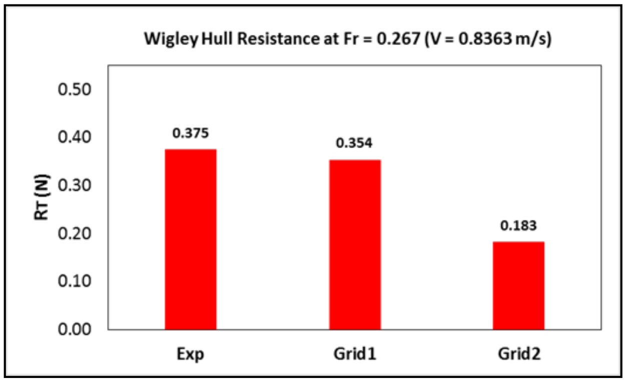

Figure 6. Comparison between Wigley hull experimental [20] and predicted numerical results at *Fr* = 0.267 (*V* = 0.8363 m/s).

The predicted wave contours by the grids around the Wigley hull at the two speeds are shown in Figs. 7 and 8. The two grids predicted nearly the same wave patterns around the hull at the two speeds. The results of Grid1 always show very sharp and good predicted wave patterns at the different speeds, while the numerical diffusion is clearly appear for all results of Grid2 due to the absence of the prism layers in the free-surface zone around the Wigley hull in the near regions.

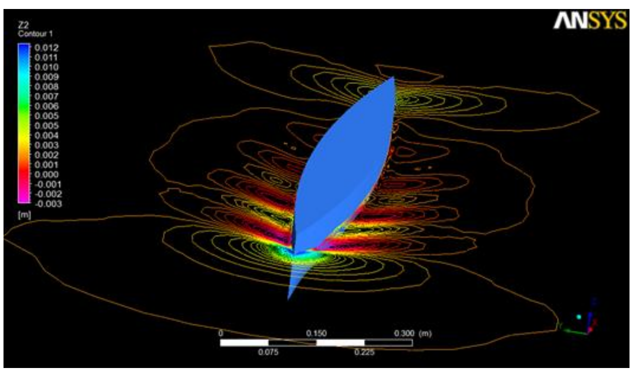

Figure 7(a). Grid1 wave contours around Wigley hull at *Fr* = 0.2 (*V* = 0.6264 m/s).

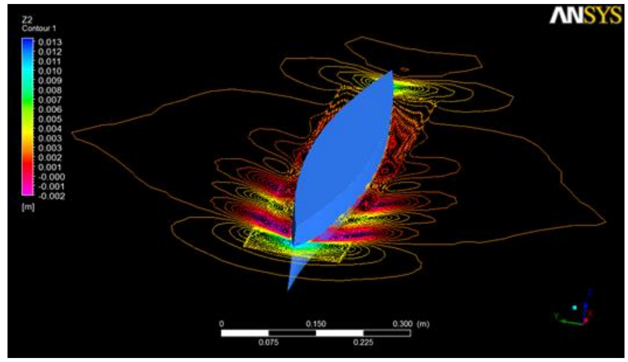

Figure 7(b). Grid2 wave contours around Wigley hull at *Fr* = 0.2 (*V* = 0.6264 m/s).

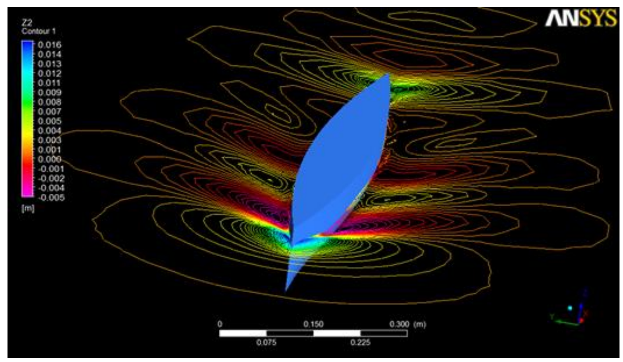

Figure 8(a). Grid1 wave contours around Wigley hull at *Fr* = 0.267 (*V* = 0.8363 m/s).

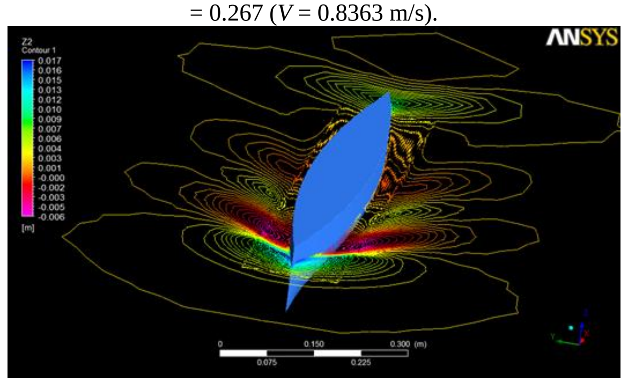

Figure 8(b). Grid2 wave contours around Wigley hull at *Fr* = 0.267 (*V* = 0.8363 m/s).

The dynamic pressure distributions on the Wigley hull at *Fr* = 0.2 by the different grids are presented in Figs. 9. The comparison between the results shows good agreement between the various results. The zones of high pressure at the bow and stern regions are detected well by Grid1 and Grid2.

The distributions of dynamic pressure for *Fr* = 0.267 (Fig. 10) are nearly similar, with some differences in the pressure amplitudes and gradients. Generally, the differences between the predicted pressure distributions by the two grids at the two speeds are due to the use of grids of different internal structure, which lead to some numerical diffusion in the predicted pressure distribution as shown in Figs. 9(b) and 10(b).

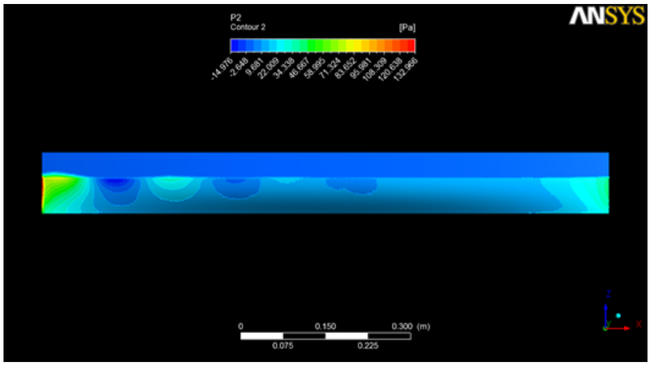

Figure 9(a). Dynamic pressure contours on Wigley hull at *Fr* = 0.2 (*V* = 0.6264 m/s) for Grid1.

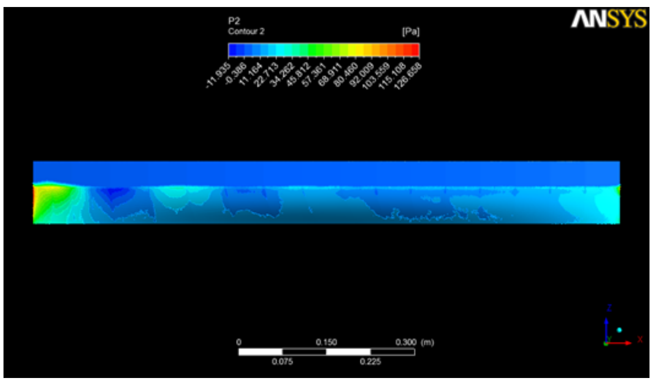

Figure 9(b). Dynamic pressure contours on the Wigley hull at *Fr* = 0.2 (*V* = 0.6264 m/s) for Grid2.

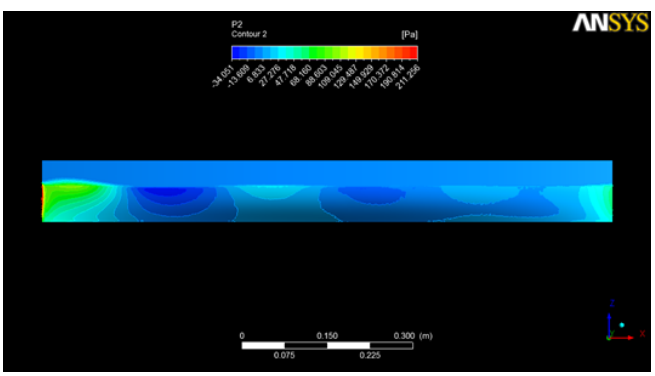

Figure 10(a). Dynamic pressure contours on the Wigley hull at *Fr* = 0.267 (*V* = 0.8363 m/s) for Grid1.

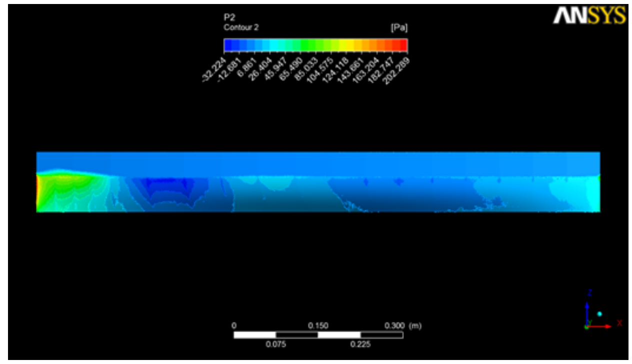

Figure 10(b). Dynamic pressure contours on the Wigley hull at *Fr* = 0.267 (*V* = 0.8363 m/s) for Grid2.

6. Conclusions

The free surface flow around Wigley hull form at two Froude numbers has been simulated using two grids with different internal structure. The finite volume code Ansys CFX has been used in this study to perform all the numerical simulations at *Fr* = 0.2 and 0.267. Grid2 gave better prediction for the resistance of the Wigley hull at *Fr* = 0.2, while Grid1 show good ability for calculation the hull resistance at the other speed.

The predicted wave contours around the Wigley hull by Grid1 are very sharp and clear at both speeds, while the results are suffering from clear numerical diffusion when the prism layers have been disappeared from the free surface zone near the ship hull area (Grid2). The pressure contours on the hull surface at different conditions of the two grids showed good agreement between the results. However, still the effect of the numerical diffusion on the contours appear in case of Grid2 results. Finally, Gird2 with large number of mesh elements shows very good advantage in the time required for the numerical simulations in comparison with Grid1of less mesh elements.

ACKNOWLEDGEMENTS

The authors would like to convey a great appreciation to Faculty of Mechanical Engineering and Marine Technology Center in Universiti Teknologi Malaysia (UTM) and Alexandria University for the great help and support.

REFERENCES

[1] Michell J. H. 1898. The wave resistance of a ship, *Philosophical Magazine*, Vol. 45: 106-123.
[2] Havelock T. H. 1928. Wave resistance, Proceedings of the Royal Society of London, Series A, Vol. 118: 24-33.
[3] Wehausen J. V. 1973. The wave resistance of ships, *Advances in Applied Mechanics*, Vol. 3: 414-420.
[4] Hess J. L. & Smith A. M. O. 1964. Calculation of nonlinear potential flow about arbitrary three-dimensional bodies, *Journal of Ship Research*, Vol. 8, No. 2: 22-44.

[5] Dawson, C. W. 1977. A Practical Computer Method for Solving Ship-Wave Problems, *Proceedings 2nd International Conference on Numerical Ship Hydrodynamics*: 30-38. Berkeley, CA.
[6] Jensen G., Soding H. 1986. Rankine methods for the solution of the steady wave resistance problem, *Proceedings 16th Symposium on Naval Hydrodynamics*: 575–582.
[7] Larson L, Broberg L, Kim KJ & Zhang DH. 1989. New viscous and inviscid CFD techniques for ship flow, *Fifth International Conference on Numerical Ship Hydrodynamics*: 1-25. Hiroshima, Japan.
[8] Gatchell S., Hafermann D., Jensen G., J. Marzi & Vogt M. 2001. Wave Resistance Computations – A Comparison of Different Approaches, *Proceedings 23rd Symposium on Navel Hydrodynamics*: 792-804.
[9] Tarafder, Md Shahjada, and Kazuo Suzuki. "Numerical calculation of free-surface potential flow around a ship using the modified Rankine source panel method." Ocean Engineering 35.5 (2008): 536-544.
[10] Hino, T. "Computation of a free surface flow around an advancing ship by the Navier-Stokes equations." International Conference on Numerical Ship Hydrodynamics, 5th. 1900.
[11] Ransau, S.R. 2002. Solution methods for incompressible viscous free surface flows: a literature review, *Report, Department of Mathematical Sciences*, Norwegian University of Science and Technology, Trondheim.
[12] Hirt, C. W. & Nichols, B. D. 1981. Volume of Fluid (VOF) Method for Dynamics of Free Boundaries, *Journal of Computational Physics*, 39, *1*: 210-225.
[13] Maronnier V. & Picasso M., Rappaz J. 1999. Numerical simulation of free surface flows, *Journal of Computational Physics*, Vol. 155: 439-455.
[14] Yong Z., Hsiang T. & Baili Z. 2002. A high-resolution characteristics-based implicit dual time-stepping VOF method for free surface flow simulation on unstructured grids, *Journal of Computational Physics*, Vol. 183: 233-273.
[15] Hino, Takanori. "Numerical simulation of a viscous flow with a free surface around a shipmodel." 日本造船学会論文集 1987.161 (1987): 1-9.
[16] Alessandrini, B., and G. Delhommeau. "Simulation of three-dimensional unsteady viscous free surface flow around a ship model." International Journal for Numerical Methods in Fluids 19.4 (1994): 321-342.
[17] Qiuxin, Gao. "Numerical simulation of free surface flow around ship hull."journal of Ship Mechanics 6.3 (2002): 1-13.
[18] Zhang, Zhi-rong, Feng Zhao, and Baiqi Li. "Numerical calculation of viscous free-surface flow about ship hull." Journal of ship mechanics 6.6 (2002): 10-17.
[19] Barth T.J. & Jesperson, D.C. 1989. The design and application of upwind schemes on unstructured meshes, *AIAA*: 89-0366.
[20] Kajitani, H et al (1983). The Summary of the Cooperative Experiments on Wigley Parabolic Model in Japan. Tokyo University Japan.