

Numerical Simulations of Viscous Flows around JBC Ship Using Different Turbulence Models

Jianhua Wang, Decheng Wan*

Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai 200240, China

*Corresponding author: dcwan@sjtu.edu.cn

ABSTRACT

High Reynold number (Re) viscous flow is one of the most typical characteristics of an advancing ship. Therefore, the wake flow after the ship hull is very complex and it will strongly affect the propulsion performance. RANS approach and hybrid LES/RANS methods are very popular in the contemporary research of ship flows, with the consideration of less computational cost compared with LES. In the present work, several numerical models, i.e. standard Shear Stress Transport (SST) $k-\omega$ model for RANS, DES, Delayed DES (DDES) and Improved Delayed DES (IDDES) model are used to predict the viscous flows around a full block ship. Numerical computations are carried out by the in-house CFD solver naoe-FOAM-SJTU developed on OpenFOAM. The ship model Japan Bulk Carrier (JBC) is used in the present computations and the numerical setup is following the benchmark case in Tokyo 2015 CFD Workshop. Extensive experimental data including detailed wake flows through PIV measurements is available for the validation of the CFD results. Predicted results of ship resistance, wake flow in different cross sections are presented and compared with the experimental data. The results based on different DES approaches are discussed and compared with the RANS results. It is found that the IDDES method can improve the performance in predicting turbulence features. Good agreement with experimental results shows that the RANS approach can give good prediction of the resistance, while IDDES approach is more suitable in predicting the complex viscous flows around full block ship hull.

Key words: Ship wake flow, JBC ship, IDDES, turbulence models

1 INTRODUCTION

With the great improvement in the computational fluid dynamics (CFD), more and more work has been done by CFD for engineering problems. In the field of ship hydrodynamics, CFD is one of the most reliable tools for the prediction of ship resistance. However, it is still very challenging to resolve the detailed flow field around ship hull, especially for full ship hull. So far, many researches have been devoted to find an accurate numerical model to resolve the high-Re viscous flows around ships. Tahara et al.[1] studied the wake flow behaviour around Series 60 ship model in steady drift motion. RANS equations with Baldwin-Lomax turbulence model was used to solve the flow field. It was concluded that the trends of integral variables and mean-flow fields can be well predicted, while the complex turbulent flows were not very good due to the turbulent models and numerical schemes. Xing et al. [2] presented numerical studies for a KVLCC2 tanker hull form using Detached Eddy Simulation (DES) on a 13 M grid. Vortical structures and turbulent structures were resolved and their results showed significantly improve on local flows. They also noted that the improved DES models will be better for the local flow predictions. Olivieri et al. [3] conducted PIV measurements of the wake flow around DTMB ship hull and numerical computations were also done using RANS approach with standard k-w turbulent model. The CFD results show fairly good agreement with the measurements, while the local flows were not accurate enough. Wang et al. [4] conducted both RANS

and DES computations for KCS ship in high Froude numbers. It was founded that the DES approach can give a better prediction of the free surface flows as well as the wake flow.

Standard shear stress transport two equation $k-\omega$ model [5,6] is widely used for engineering applications. However, the wake flows are not very satisfactory when comparing with the experiments. The detached eddy simulation (DES) model [7] was proposed to give more detailed flow features with the concept of using RANS mode for boundary layer and LES mode in large separated flow region. But the grid refinement procedure should be very careful when using the original DES model. Later on, many researches [8–10] have been devoted to make the model more effective and more reliable by proposing different kinds of models, such as the delayed DES and improved delayed-DES approach. Guilmineau et al. [11], Visonneau et al. [12] and Kornev et al. [13] investigated the turbulent features around JBC ship and they all found that the DES or Hybrid model performed better in predicting the local flows. But the simulation results depended greatly on the numerical schemes and grids.

The objective of this paper is to show the performance of different turbulence models in predicting the high-Re viscous flow around a full form ship hull. The paper is organized as follows: the numerical approaches including different turbulent models is introduced in section 2, and the geometry model, numerical grids and test conditions are presented next, then the numerical results of flow field around JBC ship is discussed in detail, finally a brief summary is given.

2 NUMERICAL APPROACH

The numerical computations are using the CFD solver naoe-FOAM-SJTU[14–16], which is developed based on the open source platform OpenFOAM. This solver has been validated on varies applications to ship hydrodynamics. Currently, the solver has the ability of solving both RANS and DES equations. The built-in modules in OpenFOAM, such as turbulence models, VOF, solver for linear system of equations make it very easy to conduct numerical simulations for different kinds of hydrodynamic problems.

In this paper, RANS and DES simulations with different kinds of turbulence models, such as standard shear stress transport (SST) $k-\omega$ model [5] for RANS (written as SST RANS), standard SST model for DES (SST DES), for delayed DES (SST DDES), for improved DDES (SST IDDES), are used to predict the high-Re viscous flow around JBC ship. The implementation of DDES and IDDES modules are following the procedure done by Gritskevich et al. [10]. The detailed parameters for different turbulence models are according to recommended value mentioned in Menter [6] and Gritskevich et al. [10]. Wall resolved approach should be used for the wake flow computations and it is better to give high accuracy predictions. But due to the high computational cost, we are using wall functions for all the calculations in the present work.

The steady flow model with local time stepping (LTS) Euler scheme is employed for the RANS computations, whereas the unsteady flow model with a second order backward scheme is used for DES calculations. Blended scheme (LUST, 25% linear upwind and 75% linear) is applied for the convection discretization and centred scheme is used for the diffusion term.

3 COMPUTATIONAL SETUP

The Japan Bulk Carrier (JBC) ship model is applied for all the present simulations. JBC is one of the benchmark ship models for CFD validation in Tokyo 2015 CFD workshop on ship hydrodynamics[17]. Detailed experimental data including PIV results are available for CFD validations. The geometry of JBC is shown in Figure 1.



Figure 1 Geometry model of JBC ship

The main particulars of the ship model are listed in Table 1.

Table 1 Main particulars of JBC ship

Main particulars	Symbol	Full scale	Model scale
Length between perpendiculars	L_{pp} (m)	280	7
Length of waterline	L_{wl} (m)	285	7.125
Beam	B (m)	45	1.125
Depth	D (m)	25	0.625
Draft	T (m)	16.5	0.4125
Displacement	$\nabla(m^3)$	178369.9	56.291
Block coefficient	C_B	0.858	0.858

All of the meshes used in this paper are generated by *snappyHexMesh*, which is a mesh generation tool provided by OpenFOAM. Given that the motion of ship is fixed, the present simulation uses a half domain as shown in Figure 2. The computational domain extends to $-1.0L_{pp} < x < 4.0L_{pp}$, $-1.25L_{pp} < y < 0$ and $-1.0L_{pp} < z < 0.5L_{pp}$. In order to resolve the local flow in the wake region, local grid refinement is used as shown in Figure 2. The y^+ is around 30 with respect to the wall functions requirement. The grid density in the near wall region should be tested in the future work. Currently, the total grid number of computational domain is 7.48 million.

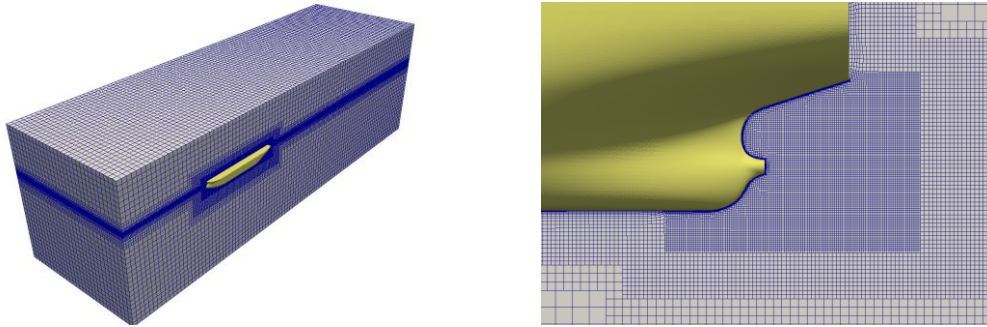


Figure 2 Computational domain and local grid distribution

The present simulations follow the experimental setup performed at NMRI and the test case is the benchmark case 1.3 in Tokyo 2015 CFD workshop. Different from the original experimental test, the ship model is fixed to the specified trim and sinkage according to the EFD data with the consideration of reducing the variables in the computation. The towing speed is 1.179 m/s corresponding to $Fr=0.142$ and $Re=7.46 \times 10^7$.

All the computations are carried out at HPC in computational marine hydrodynamics Lab (CMHL), SJTU. 60 processors are used for each case and the time step is using the adaptive steps with the Courant number limitations lower than 1.

4 RESULTS AND DISCUSSION

Four numerical models, i.e. SST RANS, SST DES, SST DDES and SST IDDES are used to predict the ship resistance and viscous flow around ship hull. Table 2 lists the comparison of ship resistance between different models and experiment.

Table 2 Comparison of ship resistance between different approaches

Numerical approaches	$C_T (\times 10^{-3})$	$C_F (\times 10^{-3})$	$C_P (\times 10^{-3})$	Error(%)
EXP.	4.289	--	--	
SST RANS	4.184	3.060	1.123	-2.458
SST DES	3.843	2.286	1.556	-10.401
SST DDES	4.140	2.920	1.220	-3.470
SST IDDES	4.061	3.023	1.037	-5.323

In Table 2, the non-dimensioned coefficients, C_T , C_F and C_P denote total resistance coefficient, friction coefficient and pressure coefficient, respectively. The coefficients are non-dimensioned by density of water (ρ), ship speed (U), and length between perpendiculars (L_{pp}):

$$C_T = \frac{R_T}{\frac{1}{2}\rho U^2 S_0} \quad (1)$$

$$C_F = \frac{R_F}{\frac{1}{2}\rho U^2 S_0} \quad (2)$$

$$C_P = \frac{R_p}{\frac{1}{2}\rho U^2 S_0} \quad (3)$$

where S_0 denotes the static wetted surface area and it can be calculated by the test data $S_0 / L_{pp}^2 = 0.2494$.

It can be seen that RANS approach can predict the integral forces very well compared with DES approaches. Among all the predicted results, the original DES approach meets the largest discrepancy, especially for the friction term. Significant drag reduction is observed and this phenomenon has been found in other studies [10,13]. It is mainly due to the ambiguous grids, which will cause grid-induced separation (GIS). The delayed DES and improved DDES approach have fixed this problem and thus the predicted ship resistance is better than the original DES results. In addition, the total resistance predicted by improved DDES approach is worse than DDES.

Apart from the integral forces discussed above, the main part of this paper is to find a reliable model in predicting the high-Re viscous flow characteristics around the ship hull. Figure 3 presents the comparison of wake flow at cross section of $X/L_{pp}=0.9625$. Both u component and w component have been shown here to illustrate the flow characters based on different methods. In each figure, left side is the PIV measurement data and the right side is the CFD results. It can be obviously seen that all the numerical models except DES can give an overall agreement prediction for the local flows. As discussed in the last paragraph, the ambiguous grid could cause the grid induced separation. Consequently, it reduces the eddy viscosity ν_t in the RANS region, which is supposed to be calculated by RANS, and the flow is easy to separate. Though the DES results reveal the existence of the closed contour familiar with PIV data, the results are not reliable since the flow is not the real physics. The w component distribution predicted by DES shows the unrealistic flow is far from the actual wake field. The IDDES approach can give a better prediction for both the u component and w component compared with other models. But the closed contour captured in the test is not well resolved.

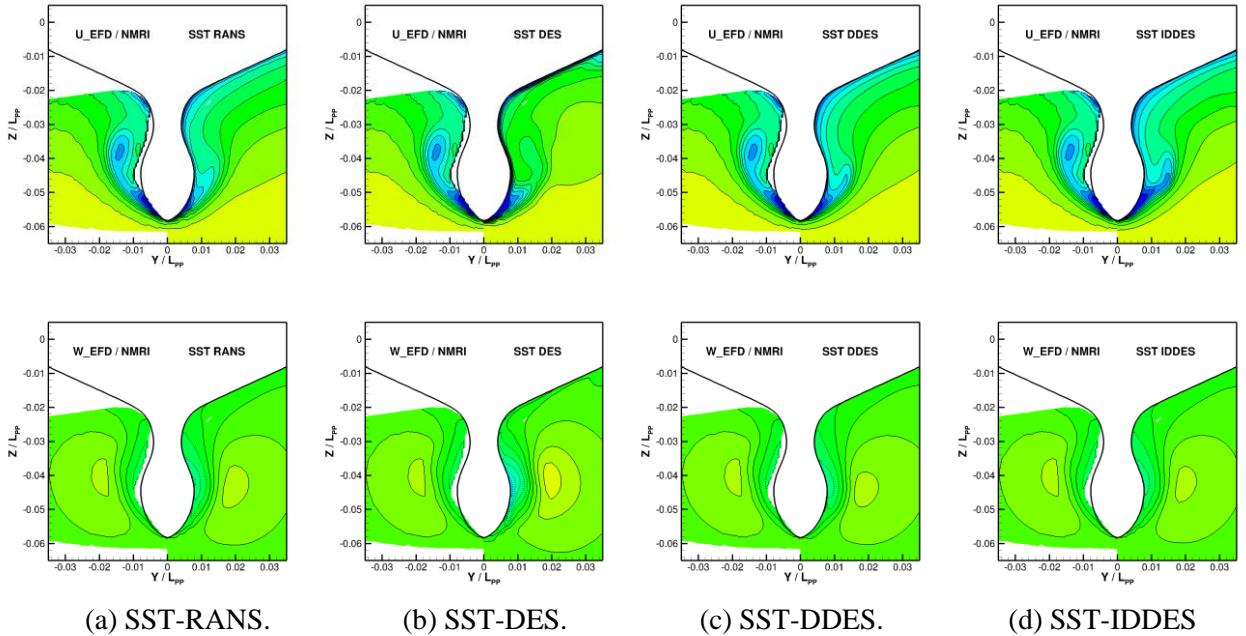


Figure 3 Comparison of wake flow at $X/L_{pp}=0.9625$ (Row #1: U component, row #2: W component)

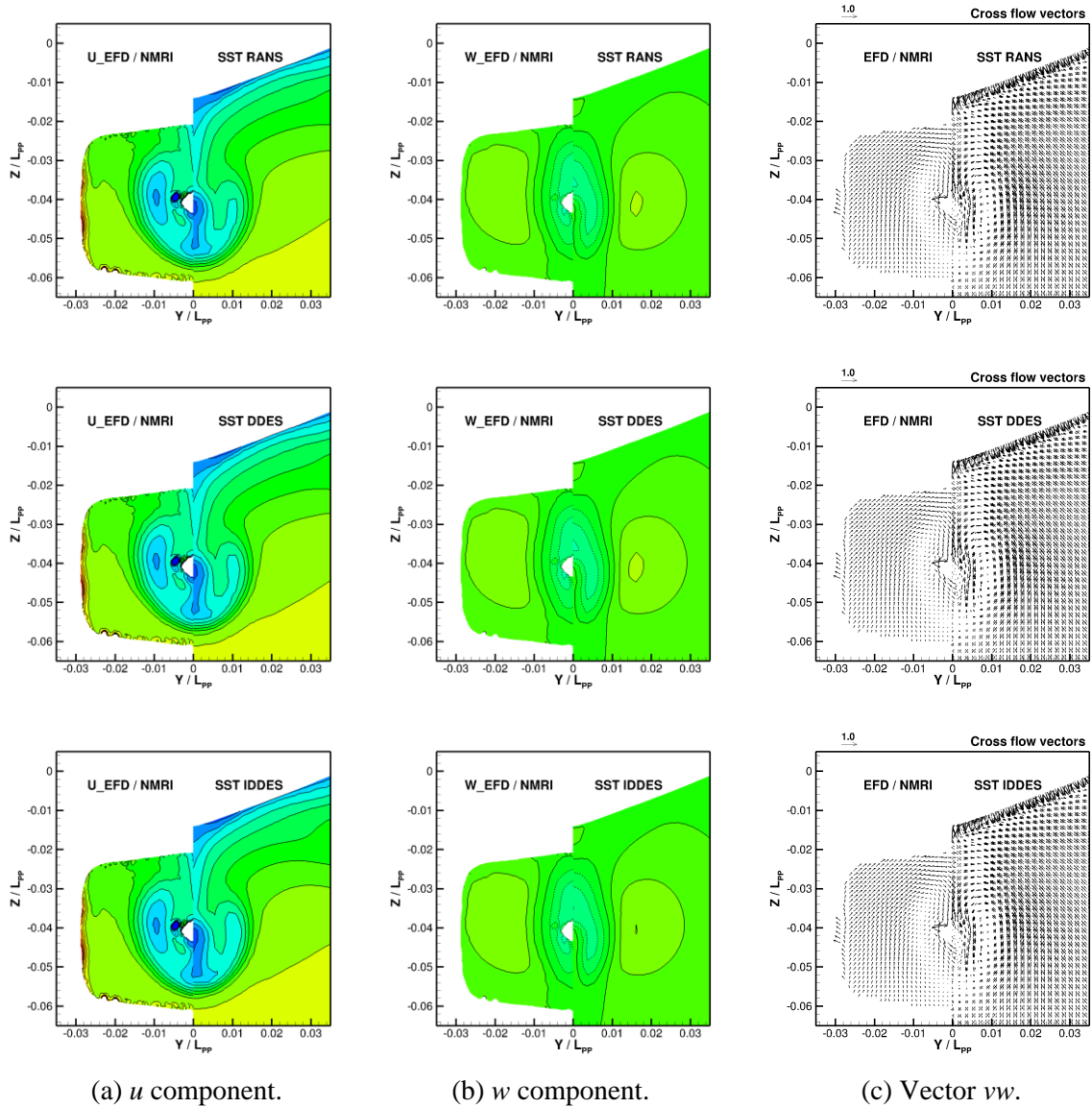


Figure 4 Comparison of wake flow at $X/L_{pp}=0.9843$ (Row #1: RANS, row #2: DDES, row #3: IDDES)

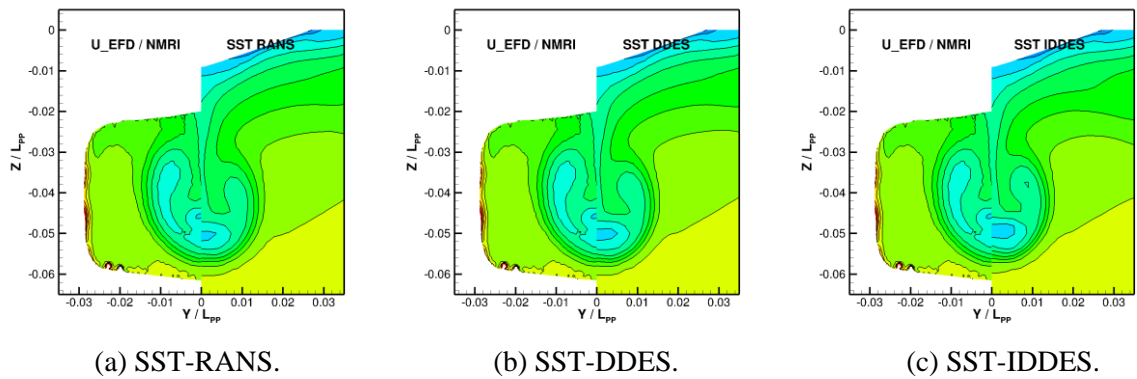


Figure 5 Comparison of wake flow at $X/L_{pp}=1.0$

Figure 4 demonstrates the comparison of wake flow at cross section $X/L_{pp}=0.9843$ and three components including u , w and vector field νw are presented to give better description of the flow features compared with PIV measurements. It is obvious that the IDDES has a better performance when predicting the existing of closed contours. This can be understood better by the view of w component. Figure 5 depicts

the comparison of wake flow at $X/L_{pp}=1.0$. It can be noticed that the IDDES approach can well resolve the local flow features, while it still has some discrepancies compared with measurements. According to the state-of-the-art study of ship wake flow using hybrid LES/RANS approach, our present grid is too coarse to get desired viscous flow details. On the whole, the IDDES can resolve more turbulent flow features than RANS, DES and DDES approach.

5 CONCLUSION

In the present paper, the high-Re viscous flows around JBC hull have been calculated using CFD solver naoe-FOAM-SJTU with different turbulence models. The predicted ship resistance including total resistance, friction drag and pressure drag have been presented and compared with experimental results. The RANS approach with standard SST $k-\omega$ turbulence model gives the most accurate result than other DES models and the same conclusion has also been draw by other researchers [13]. As for the wake flows around ship hull, the IDDES approach is the best choice since it can resolve relatively smaller flow features. But IDDES module is very complex and more attention should be paid on the proper numerical schemes as well as the grid arrangement to get better results. Actually, the wake flow is very hard to give an accurate prediction especially for full form ships. Wall resolved hybrid model with finer grids can be better when conducting simulation of the high-Re viscous flows around full ship hull. However, it is still challenging to give very accurate prediction of both resistance and wake flow using hybrid LES/RANS methods.

ACKNOWLEDGEMENTS

This work is supported by the National Natural Science Foundation of China (51809169, 51879159, 51490675, 11432009, 51579145), Chang Jiang Scholars Program (T2014099), Shanghai Excellent Academic Leaders Program (17XD1402300), Program for Professor of Special Appointment (Eastern Scholar) at Shanghai Institutions of Higher Learning (2013022), Innovative Special Project of Numerical Tank of Ministry of Industry and Information Technology of China (2016-23/09), to which the authors are most grateful.

REFERENCES

- [1] Tahara, Y., Longo, J., and Stern, F., 2002, "Comparison of CFD and EFD for the Series 60 C B= 0.6 in Steady Drift Motion," *J. Mar. Sci. Technol.*, **7**(1), pp. 17–30.
- [2] Xing, T., Bhushan, S., and Stern, F., 2012, "Vortical and Turbulent Structures for KVLCC2 at Drift Angle 0, 12, and 30 Degrees," *Ocean Eng.*, **55**, pp. 23–43.
- [3] Olivieri, A., Pistani, F., Wilson, R., Campana, E. F., and Stern, F., 2007, "Scars and Vortices Induced by Ship Bow and Shoulder Wave Breaking," *J. Fluids Eng.*, **129**(11), pp. 1445–1459.
- [4] Wang, J., Ren, Z., and Wan, D.C., 2018, "RANS and DDES Computations of High Speed KRISO Container Ship," *Proceedings of the 32nd Symposium on Naval Hydrodynamics*, Hamburg, Germany.
- [5] Menter, F. R., 1994, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA J.*, **32**(8), pp. 1598–1605.
- [6] Menter, F. R., Kuntz, M., and Langtry, R., 2003, "Ten Years of Industrial Experience with the SST Turbulence Model," *Turbul. Heat Mass Transf.*, **4**(1), pp. 625–632.
- [7] Spalart, P., Jou, W., Strelets, M., and Allmaras, S., 1997, "Comments of Feasibility of LES for Wings, and on a Hybrid {RANS/LES} Approach."
- [8] Spalart, P. R., Deck, S., Shur, M. L., Squires, K. D., Strelets, M. K., and Travin, A., 2006, "A New Version of Detached-Eddy Simulation, Resistant to Ambiguous Grid Densities," *Theor. Comput. Fluid Dyn.*, **20**(3), pp. 181–195.
- [9] Spalart, P. R., 2009, "Detached-Eddy Simulation," *Annu. Rev. Fluid Mech.*, **41**(1), pp. 181–202.
- [10] Gritskevich, M. S., Garbaruk, A. V., Schütze, J., and Menter, F. R., 2012, "Development of DDES and IDDES Formulations for the K- ω Shear Stress Transport Model," *Flow Turbul. Combust.*, **88**(3), pp. 431–449.
- [11] Guilmineau, E., Deng, G. B., Queutey, P., Visonneau, M., and Wackers, J., 2019, "Detached Eddy Simulations of the Flow Around the Japan Bulk Carrier (JBC)," *Direct and Large-Eddy Simulation XI*, M.V. Salvetti, V. Armenio, J. Fröhlich, B.J. Geurts, and H. Kuerten, eds., Springer International Publishing, pp. 579–585.
- [12] Visonneau, M., Deng, G., Guilmineau, E., Queutey, P., and Wackers, J., 2016, "Local and Global Assessment of the Flow around the Japan Bulk Carrier with and without Energy Saving Devices at Model and Full Scale," *Proceeding 31th Symposium on Naval Hydrodynamics*, Monterey, USA.

- [13] Kornev, N., Shevchuk, I., Abbas, N., Anschau, P., and Samarbakhsh, S., 2019, “Potential and Limitations of Scale Resolved Simulations for Ship Hydrodynamics Applications,” *Ship Technol. Res.*, **66**(2), pp. 83–96.
- [14] Wang, J., Zhao, W., and Wan, D.C., 2019, “Development of Naoe-FOAM-SJTU Solver Based on OpenFOAM for Marine Hydrodynamics,” *J. Hydrodyn.*, **31**(1), pp. 1–20.
- [15] Shen, Z., Wan, D.C., and Carrica, P. M., 2015, “Dynamic Overset Grids in OpenFOAM with Application to KCS Self-Propulsion and Maneuvering,” *Ocean Eng.*, **108**, pp. 287–306.
- [16] Zhao, W., Zou, L., Wan, D.C., and Hu, Z., 2018, “Numerical Investigation of Vortex-Induced Motions of a Paired-Column Semi-Submersible in Currents,” *Ocean Eng.*, **164**, pp. 272–283.
- [17] Larsson, L., STERN, F., Visonneau, M., Hirata, N., Hino, T., and Kim, J., 2015, *Tokyo 2015 – a Workshop on CFD in Ship Hydrodynamics*.