Proceedings of 8th International Conference on Mechanical, Industrial and Energy Engineering 2024
02-04 January, 2025, Khulna, BANGLADESH
SciEn Conference Series: Engineering Vol. 3, 2025, pp 102-107 https://doi.org/10.38032/scse.2025.3.22

Computational Fluid Dynamic Analysis of Energy Saving Device of a Bulk Carrier

*Sayed Sadik Siddique**, *Md. Mashiur Rahaman*

Department of Naval Architecture and Marine Engineering, Bangladesh University of Engineering and Technology, Dhaka-1000, Bangladesh

ABSTRACT
With the increasing concern of the International Maritime Organization (IMO) regarding greenhouse gas emissions, the marine industry is now looking to improve ship performance by introducing various energy-saving technologies. In the present study, Computational Fluid Dynamics analysis is conducted on the test case of the JBC (Japan Bulk Carrier) model introduced in the Tokyo Workshop 2015 on ship hydrodynamics. The test case is available in a model scale of 1:40. Two variants of the model, with and without an Energy Saving Device (ESD) known as a wake-equalizing duct, are studied at Froude number 0.142 towed in calm water. An open-source RaNS-based CFD solver, OpenFOAM 11, is used for all numerical simulations. The results are validated against experimental data provided by the NMRI (National Maritime Research Institute). The numerical results show a close approximation to the experimental results. A systematic verification study is also performed based on grid size to assess numerical uncertainty. The effect of the wake equalizing duct is to reduce the total resistance by 0.5202% which is good enough for bulk carrier ships

Keywords: CFD, JBC, ESD, Resistance, OpenFOAM.

1. Introduction

Restrictive environmental controls have been imposed on the shipbuilding sector and maritime transportation due to the increasing effects of global warming and the depletion of natural resources. A primary goal for shipowners and designers is to maximize a vessel's efficiency while at sea. Optimizing propulsion systems to attain higher performance or reducing resistance are the possible solutions. To accomplish this, energy-saving devices, or ESDs, are now manufactured in substantial numbers and positioned in front of, behind, or directly on the propellers of commercial vessels. Energy-saving devices (ESDs) have been developed increasingly due to growing concerns about the environment and energy efficiency in recent decades. One type of energy-saving device investigated in this paper is the wake equalization duct (WED), which is positioned ahead of the propeller to improve propulsion efficiency and propeller-induced vibrations. As per the experimental results presented at the Tokyo workshop in 2015 [1], the Japan Bulk Carrier (JBC) equipped with WED proved enhanced propulsion efficiency and decreased resistance than the bare hull without ESD. Numerous techniques, including computational fluid dynamics (CFD), numerical methods, and experimental model tests, can be employed to predict the efficiency of energy-saving devices.

However, the use of Computational Fluid Dynamics (CFD) has increased due to the rapid development of numerical methods and CFD codes in recent years. Its fine representation of flow fields which is essential for assessing the effects of energy-saving devices—and its physics-based modeling, along with its capacity to handle non-linear free surfaces, are its strengths. Maasch et al. [2] developed a Wake Analysis Tool (WAT) for automated CFD integration, effectively analyzing the propeller wake flow of the Japan Bulk Carrier (JBC). According to Yin et al. [3] by CFD studies and experiments, the wake equalizing duct (WED) can greatly increase the propulsion efficiency of the Japan Bulk Carrier (JBC), up to 6.7%. The analysis of Chen Wu et al. [4] indicates that the Japan Bulk Carrier (JBC) can use the ideal 7° stern duct to cause a reduction in resistance up to 2.49%. Their CFD result demonstrated that improved performance can be obtained above the original duct. The study of Sun et al. [5] confirms resistance, sinkage, trim, and local flow around the Japan Bulk Carrier's stern are all accurately predicted by the naoe-FOAM-SJTU solver both with and without the wake equalizing duct.

From, the above literature review it is seen that most of the studies are conducted based on commercial or in-house CFD codes. The present study employs an open-source solver OpenFOAM to perform numerical simulation and compare the resistance of a Japan Bulk Carrier, both with and without an energy-saving device. The numerical simulation includes computations for ship resistance, with two degrees of freedom heave and pitch. A Verification and Validation (V&V) study is performed to evaluate the modeling errors that may arise in the computation.

*Corresponding Author Email Address: syed.sadik.siddique@gmail.com

2. Computational methods

An open source RANS-based CFD solver OpenFOAM version 11 is used in this present study. The numerical model used in the OpenFOAM solver is based on the continuity equation (1) and the Navier-Stokes or momentum equation (2) for incompressible flow.

$$\nabla . v = 0 \tag{1}$$

$$\rho\left(\frac{\partial u}{\partial t} + u.\nabla u\right) = -\nabla p + \mu \nabla^2 u + \rho g \tag{2}$$

In OpenFOAM, the Volume of Fluid (VOF) method is employed to capture the free surface, with the volume fraction determined by equation (3).

$$\frac{\partial \alpha}{\partial t} + \nabla(\alpha v) = 0 \tag{3}$$

For simulating the free surface, pressure-velocity coupling was achieved using the PIMPLE algorithm, which combines of both SIMPLE and PISO. For turbulence energy and dissipation rate, two equations from the SST k-omega turbulence model have been used.

3. Boundary condition

The computational domain and boundary condition for half domain (half hull) calm water simulation are shown in Fig.1 and Table 1 respectively.

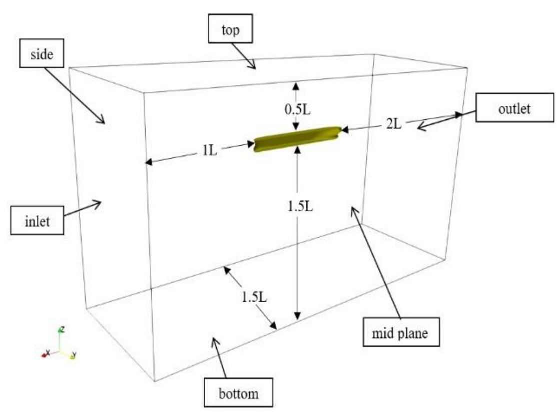


Fig.1 3D view of Computational domain

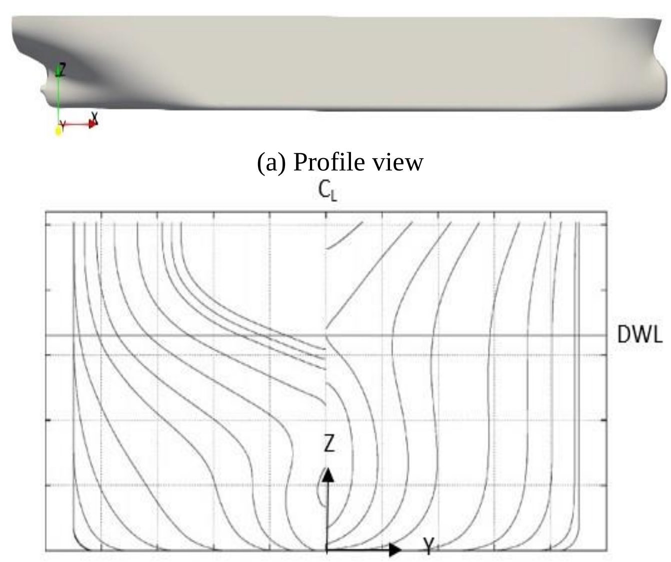
Table 1 Boundary condition in calm water condition.

Boundary	Velocity (ms^{-1})	Pressure ($kgm^{-1}s^{-2}$)
Inlet	fixedValue	fixedFluxPressure,
Outlet	outletPhaseMeanVelocity	zeroGradiendt
Hull	movingWallVelocity	zeroGradient
Sides	symmetryPlane	symmetryPlane
Bottom	symmetryPlane	symmetryPlane
Top	pressureInletOutletVelocity	totalPressure

Here, fixedValue means a constant value specified by the user end input, in our case it is (-1.179,0,0) zeroGradient means no change with respect to time, fixedFluxPressure sets the gradient of pressure to the specified by the flux on the velocity boundary condition which is equal to (0) in the present study, pressureInletOutletVelocity specifies velocity where pressure is defined and here it is (0,0,0) outletPhaseMeanVelocity adjusts the velocity such that causing the phase-fraction to adjust according to the mass flow rate which is (1.179,0,0) here. movingWallVelocity is the boundary condition for a wall-type object moving with a constant velocity and it is (0,0,0) in the present study, symmetryPlane specifies no interaction with fluid flow. Boundary conditions for ESD are the same as Hull. Details about these boundary conditions are given in OpenFOAM user guide.

4. Ship geometry

The JAPAN Bulk Carrier (JBC) is a Capesize bulk carrier developed by the National Maritime Research Institute (NMRI), Yokohama University, and the Shipbuilding Research Centre of Japan (SRC), with assistance from ClassNK (Nippon Kaiji Kyokai). The vessel is outfitted with a wake equalizing duct, which functions as an energy-saving device. In this paper, JBC hull without and with energy-saving device (ESD) have been taken into account in model scale with the scale factor of 40. The profile view and body plan of ship model have been shown in Fig.2 and the principal particulars are presented in Table 2.



(b) Body plan

Fig.2 (a) Profile view and (b) body plan of JBC hull

Table 2 Principal Particulars of JBC Hull

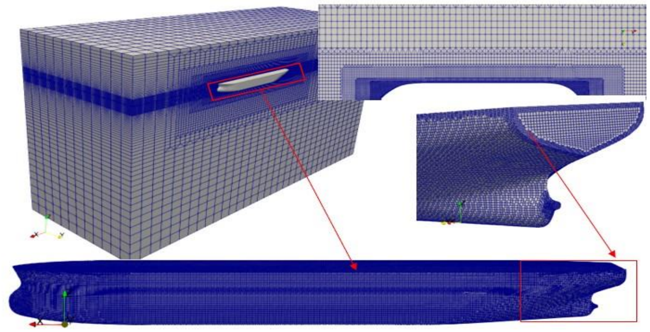
Specification	Symbol	Full scale	Model scale	Unit
Length between perpendicular	L_{pp}	280	7	[m]
Length waterline	L_{WL}	285	7.125	[m]
Beam	B_{WL}	45	1.125	[m]
Depth	D	24	0.625	[m]
Draft	T	16.5	0.4125	[m]
Displacement volume	∇	178369.9	2.7870	[m^3]
Wetted surface area without ESD	$S_{w/oESD}$	19556.1	12.222	[m^2]
Wetted surface area with ESD	S_{wESD}	19633.9	12.270	[m^2]
Block co-efficient	C_B	0.858	0.858	--
Mid ship section co-efficient	C_M	0.9981	0.9981	--
Longitudinal center of buoyancy	LCB % Lpp	2.5475	2.5475	--
Scale factor	λ	1	40	--

5. Case condition

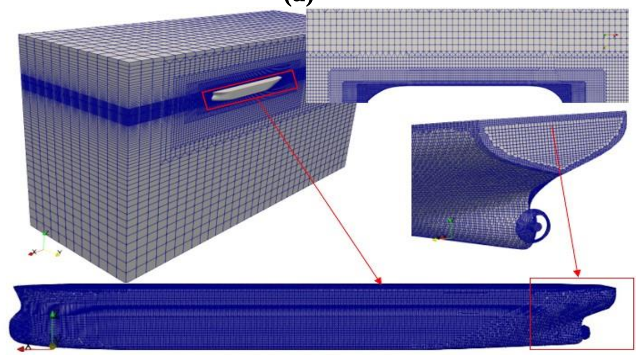
JBC models at a Reynolds number of Re= 7.46×10^6 and a Froude number of Fr no. = 0.142, corresponding to a velocity of 1.179 [ms^{-1}] in model scale (14.5 knot in full scale) are considered in the current study. To facilitate a comparison between the numerical simulations and the experimental results, the case conditions are taken same as model tests. The density of water is taken as 998.2 kg/m^3, with a kinematic viscosity of 1.107×10^{-6} [$m^2 s^{-1}$]. The gravitational acceleration was taken as 9.80 ms^{-2}. All the simulations are run at heave and pitch free motion up to 60 seconds (simulation time). A desktop computer running Linux OS powered with Intel Core-i7 (10700) processor with 16 GB of DDR-4 RAM has been used for computational operation.

6. Mesh generation

In OpenFOAM, the blockMeshDict dictionary is used to create the computational domain according to the ITTC 2011 guidelines. The setup includes two ship length behind the aft perpendicular, one ship length in front of the forward perpendicular, one and a half ship lengths from the centerline to the side, one and a half ship lengths from the free surface to the bottom, and half a ship length from the free surface to the top side. OpenFOAM's built-in mesh generation utility, snappyHexMesh dictionary, is used to generate the mesh around the hull. The overall domain mesh along with detailed near the stern is shown in Fig. 3. Successive refinements have been done using six topoSetDict dictionary. The grid resolution near the bow and stern (duct) has been improved to effectively capture the free surface and wake flow field, while also enabling precise calculations in the wall region of the hulls, both with and without ESD. Cell numbers for simulation without and with ESD are 2.238 and 2.305 million respectively.



(a)



(b)

Fig.3 General computational mesh assembly for JBC hull (a) without and (b) with ESD

7. Verification study

For verification study, bare hull without ESD has been considered. Three different meshes with a uniform refinement ratio 1.18 has been used for the verification study. The two most popular methods used to analyze uncertainties are the Grid Convergence Index (GCI) with Factor of Safety and Correction factor (C_i). The GCI with Factor of Safety approach was described well by Celik et al. [6] . The GCI method estimates uncertainty from grid and time step errors using Richardson extrapolation with multiple solutions on refined grids.. Stern et al. [7] proposed a correction factor-based approach. Correction factors approximate the impacts of higher-order terms in error-making and offer an approach for quantifying the distance of solutions from the asymptotic range. For uncorrected solutions, the uncertainty estimation is derived from the absolute value of the corrected error estimate combined with the correction amount. In corrected solutions, the corrected error estimate, considering both sign and magnitude, establishes a numerical benchmark based on the absolute value of the correction amount. Here, resistance co-efficient (C_T) and trim show monotonic convergence whereas sinkage shows oscillatory convergence shown in Table 3.

8. Results and discussion

In the current study, two models of JBC hull without and with ESD have been considered. Numerical simulations of both hulls are performed at Froude no 0.142 with 2 DOF (Heave and Pitch). The numerical results are compared with experimental results provided by NMRI (National Maritime Research Institute) in Tokyo CFD Workshop 2015. After validation is done the comparison is made of numerical results without and with ESD.

8.1 Validation of numerical results

Validation of the results of total resistance co-efficient (C_T), trim and sinkage of JBC hull without and with ESD against experimental results are mentioned in Table 4 and Table 5 respectively.

Table 3 Verification study

Parameter	Cell No. (million)	C_T (10^{-3})	Trim (deg)	Sinkage (Z/Lpp%)
Mesh 1 (Fine)	2.23	4.421	-0.192	-0.093
Mesh 2 (Medium)	1.46	4.480	-0.193	-0.094
Mesh 3 (Coarse)	0.98	4.573	-0.1938	-0.089
Grid Convergence Index $GCI_{(21)}$	--	0.030	0.0117	0.009
Grid Convergence Index $GCI_{(32)}$	--	0.047	0.0153	0.0086
Corrected Uncertainty $U_{(1C)}$	--	0.99%	0.18	0.149
Corrected Uncertainty $U_{(2C)}$	--	1.51%	0.24	13.51

Table 4 Validation of results without ESD

Results	$C_T(10^{-3})$	Trim (% Lpp)	Sinkage (% Lpp)
EFD	4.289	-0.18	-0.086
CFD	4.421	-0.192	-0.093
% Error	3.09	6.933	9.285

Table 5 Validation of results with ESD

Results	$C_T(10^{-3})$	Trim (% Lpp)	Sinkage (% Lpp)
EFD	4.263	-0.185	-0.085
CFD	4.398	-0.196	-0.094
% Error	3.17	6.165	11.68

From Table 4 and Table 5 it is seen that the numerical results show a very close approximation to experimental results except for trim and sinkage where the errors are beyond 5%. The possible reason is an inappropriate selection of COG (center of gravity). The location of COG is not known in experimental conditions. The results may be improved by increasing cell no in z- direction.

8.2 Comparison of results

Comparison of both hull without and with ESD device is made on viscos and pressure resistance mentioned in Table 6. From Table 6, it is seen that the effect of ESD (Wake Equalizing Duct) is to reduce the viscous (frictional) resistance and increase the pressure (form) resistance. However, the decrement in viscous resistance should be large enough to counter the increment in pressure resistance. So, the net effect will be a decrease in total resistance co-efficient C_T.

Table 6 Comparison of results WO/ESD and W/ESD

Results	C_p (10^{-3})	$C_v(10^{-3})$	C_T (10^{-3})
WO/ESD	1.303	3.117	4.421
W/ESD	1.304	3.095	4.398
% of Reduction	-0.0767	0.7058	0.5202

The main function of a wake equalizing duct (ESD) is to modify the flow around the hull. It accelerates the flow and smoothens the flow separation near the stern. This leads to an increase the form drag and a decrease of viscous drag. To show how sensitive the ESD is to total resistance co-efficient C_T. Comparison of net decrement of resistance (%) between EFD and CFD is shown in Fig 4.

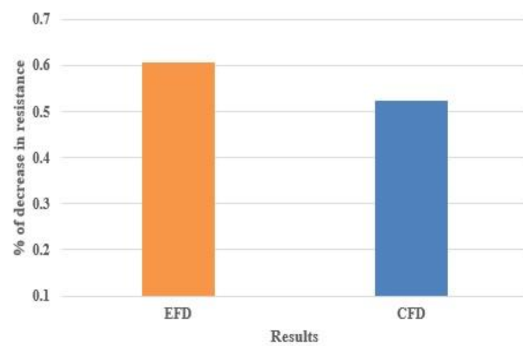


Fig.4 Bar chart showing comparison of reduction in total resistance co-efficient C_T between experimental and numerical results.

The percentage of decrease in total resistance coefficient in experimental and numerical results are 0.602 % and 0.5202 % respectively which are close enough.

9. Analysis of the flow field

The Kelvin wave pattern, wakefield, streamline and distribution of pressure on the hull are analyzed in the present study.

The wave pattern produced by the hull in calm water plays a crucial role in influencing ship resistance, particularly in terms of wave-making resistance. Fig. 5 shows the free surface elevation of the JBC hull without and with ESD.

The wave pattern without ESD shows pronounced Kelvin waves with larger amplitudes, increasing wave-making resistance. On contrast, with an ESD, the wave pattern is modified and exhibits reduced wave heights is shown in Fig 5. Generally, a wake equalizing duct reduces disturbance near the stern making the flow more streamlined shown in Fig 6. This causes lower wave-making resistance when ship is equipped with ESD. Wake-field near the stern at 1.57 % of Lpp is shown below in Fig 7.

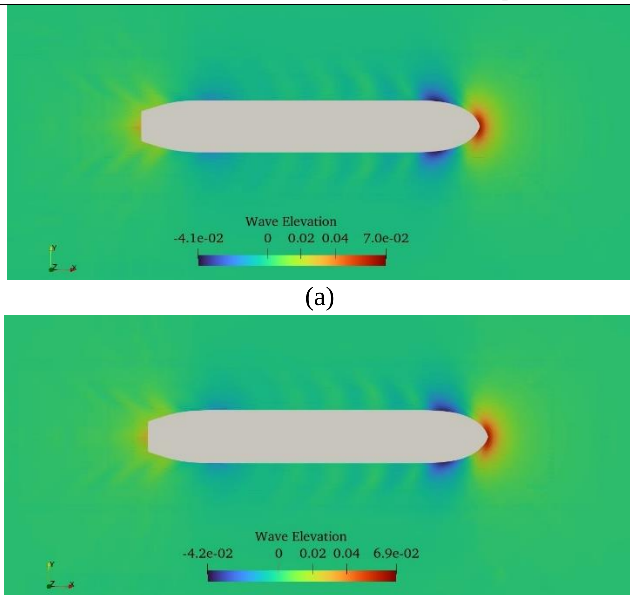


Fig.5 Free surface wave elevation (a) JBC hull without ESD (b) JBC hull with ESD

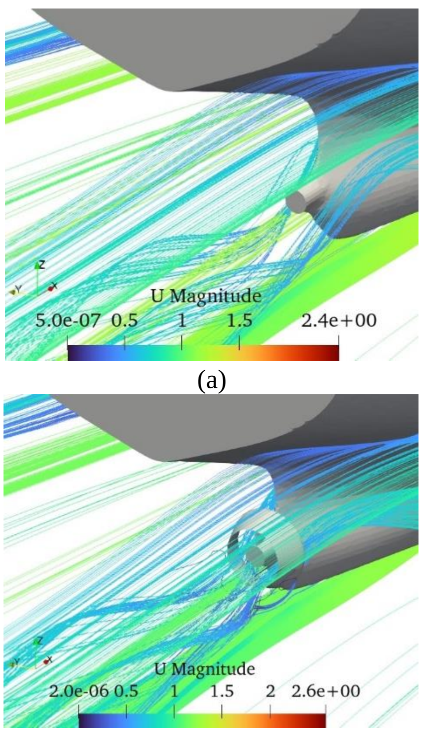


Fig.6 Streamline near the stern (a) JBC hull without ESD (b) JBC hull with ESD.

With ESD the wakefield looks more symmetric and uniform (equalized) is illustrated in Fig 7. It makes the flow separation near the stern smoother and reduces turbulence providing better in flow velocity to the propeller plane. The hydrodynamic pressure field over the hull remains the same without and with ESD except for pressure near the stern in Fig. 8.

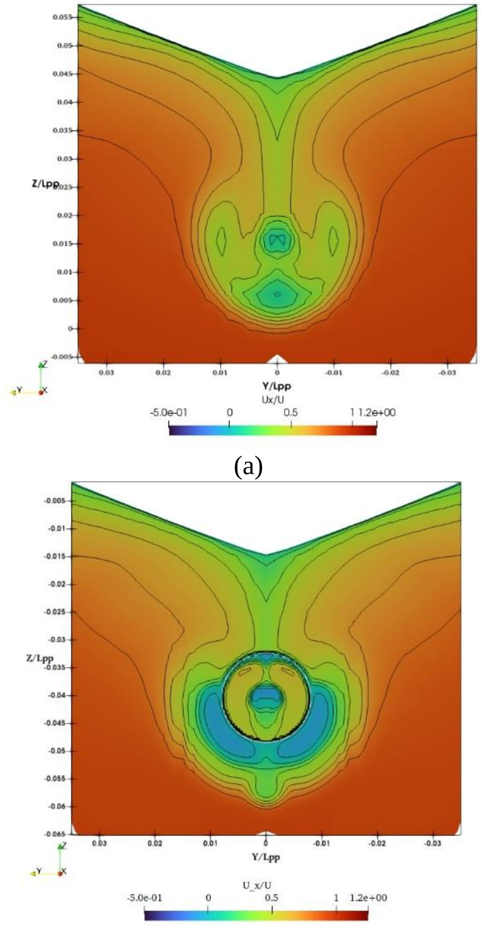


Fig.7 Wakefield (a) JBC hull without ESD (b) JBC hull with ESD

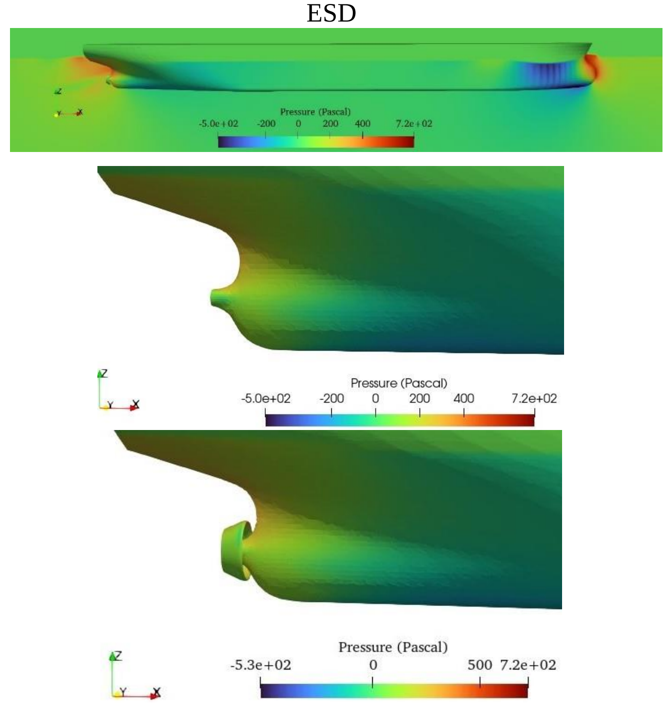


Fig.8 Distribution of hydrodynamic pressure on hull, near the stern and ESD

The local pressure near the stern increases due to the presence of a wake equalizing duct. It accelerates the flow to the propeller causing an acceleration to local flow.

10. Conclusion

The paper has investigated the effect of energy saving device (ESD) on bare hull performance of a bulk carrier sailing in calm water. The study provides a valuable insight of a wake equalizing duct by reducing total resistance of ships. The resistance components, trim and sinkage are compared to the model test results. The validation study shows good agreement with experimental results. In resistance prediction, the error is not more than 5% with the experimental result.

In the present study, a wake equalizing duct made of NACA- 4402 section has been implemented. The purpose of the wake equalizing duct is to increase the inflow velocity to the propeller plane and reduce the losses in the wake. The total resistance coefficient is reduced by 0.5202 % by implementing the wake equalizing duct, which is large enough for such a cape-size bulk carrier. However, the present study may be a good source to optimize vessel performance reduction in greenhouse and operating costs by implementing a wake equalizing duct.

The study is conducted with limited mesh resolution. Future recommendation is to increase the mesh density around the duct. This may provide closer result to experiment.

References

[1] Hino, T., Stern, F., Larsson, L., Visonneau, M., Hirata, N., Kim, J., Numerical Ship Hydrodynamics: An Assessment of the Tokyo 2015 Workshop, Dordrecht, The Netherlands: Springer, 2020.

[2] Maasch, M., Mizzi, K., Atlar, M., Fitzsimmons, P., Turan, O., A generic wake analysis tool and its application to the Japan Bulk Carrier test case, *Ocean Engineering*, vol. 171, pp. 575-589, 2019.

[3] Yin, C., Wu, J., Sun, T., Wan, D., A NUMERICAL STUDY FOR SELF-PROPELLED JBC WITH AND WITHOUT ENERGY SAVING DEVICE, *Engineering, Environmental Science*, 2016.

[4] Wu, P., Chang, C., Huang, Y., Design of Energy-Saving Duct for JBC to Reduce Ship Resistance by CFD Method, *Energies*, vol. 15, no. 17, 2022.

[5] Sun, T., Yin, C., Wu, J., Wan, D., Numerical Computations of Resistance for Japan Bulk Carrier in Calm Water, *Engineering, Environmental Science*, 2016.

[6] Celik, I., Ghia, U., Roache, P.J., Freitas, C.J., Coloman, H., Raad, P.E., Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications, *Journal of Fluids Engineering*, vol. 130, no. 7, p. 078001(4 pages), 2008.

[7] Stern, F., Wilson, R., Shao, J., Quantitive V&V of CFD simulations and certification of CFD codes, *INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS* , vol. 50, no. 11, pp. 1335-1355, 2006.