



International Journal of Science, Architecture, Technology and Environment

Numerical Study on Open Water Propulsion for Gate Rudder Application – A Case Study of DTMB 5415

Yusuf Ahmad Maidala^{1*}

¹*Department of Naval Architecture and Marine Engineering Piri Reis University Istanbul, Turkiye.*

^{*}*Corresponding author*

DOI: <https://doi.org/10.63680/ijstate0925043.30>

Abstract

Maritime transport has continued to play a major role in the transport of goods and serves across the globe. Therefore, optimizing vessel performance will continue to be a subject of study in the maritime space for enhanced efficiency. However, the Dynamics of fluid flow in the field of Naval Architecture have continued to challenge hydrodynamist in analyzing complex flow problem around hulls of vessels particularly involving a novel Energy Saving Device (ESD) such as Gate rudder. The study began with choosing a suitable twin screw naval ship with available benchmark result, hence, the choice of a DTMB 5415. The bare hull of DTMB was simulated for resistance validation using a multiphase volume of fluid as well as an open water propeller propeller simulation for the vessel. The model used for the study was k- ϵ model. The result showed that it was possible to validate the resistance of a DTMB 5415 bare hull with the model. Thereafter a conceptual Gate Rudder System (GRS) design is adopted for suitable for Naval Operations. The numerical analyses in the study was performed using the Reynolds Averaged Navier Stokes (RANS) model in the ANSYS Fluent computational fluid dynamics (CFD) software.

Keywords: Naval Ship, Gate Rudder, Propeller, RANS, CFD.

Nomenclature

CFD	: Computational Fluid Dynamics
DTMB	: David Taylor Model Basin
EEXI	: Energy Efficiency Existing Ship Index
ESD	: Energy Saving Devices
GCI	: Grid Convergence Index
GHG	: Green House Gas
GRS	: Gate Rudder System
IMO	: International Maritime Organization
ITTC	: International Towing Tank Committee
MRF	: Multiple Reference Frame
NACA	: National Advisory Committee for Aeronautics
NSE	: Navier-Stokes Equations

1. Introduction

The application of Energy Saving Devices (ESD) in the maritime sector to help ameliorate the effect of climate change which is caused by global warming has never been this dire as a necessity. In 2018, the International Maritime Organization (IMO) noted that commercial shipping industry was said to be responsible for about 2.89% of Green House Gas (GHG) emissions in the global maritime space (Opdam, 2024). The organization's strategy on GHG is said to envisage that, in particular, a reduction in CO₂ emissions which is from international shipping by at least 40% by 2030 (IMO, 2023). One of the strategies to be employed by the organization to reduce the intensity of carbon is by striving to reduce 10% of Energy Efficiency Existing Ship Index (EEXI) for international shipping by 2030. Thus EEXI could be achieved by leveraging on ship Energy Efficiency Design Index (EEDI) which requires the continuous improvements in ship efficiency through design effort (IMO, 2015). Hence, the need for improved energy efficiency for vessels by using Energy Saving Devices (ESD) such as pre-swirlers. Other ESD such as GRS could be applied to reduce both GHG, noise level as well as the amount of energy used onboard vessels. See Figure 1, a picture of a Gate rudder.



Fig 1. Picture of Gate Rudder source: Komane propeller.co

GRS is a relatively new ESD that have revolutionised the arrangement of ships' propeller and rudder. The propeller rudder arrangements in the system is such that two rudder blades with has asymmetric sections that are placed adjacent to the propeller of the ship where independent control of the blade (Sasaki et al., 2018). The arrangement is that, the two rudder blades creates a duct effect similar to ducted propeller by encircling the propeller at the top and sides. It is believed that the arrangement produces more thrust than a conventional rudder behind the propeller that creates additional drag of (Sasaki et al., 2018). The system was invented by (Kuribayashi and Sasaki, 2012), and the noted potential energy saving capabilities of the GRS was first reported by Sasaki in 2013 (Sasaki et al., 2018).

The energy saving capability of a GRS system was first revealed from voyage data recorded from two 2400

dwt container ships, in a full scale performance. The results gotten with one that was equipped with the world's first GRS has shown 14% reduction in fuel consumption (Sasaki et al., 2019). Other researches on GRS include the recently organized H2020 project conducted by GATERS which commenced in 2020 which aims to leverage the application of the novel propulsion and manoeuvring device. The GATERS project designed, manufactured and installed a GRS onboard a general cargo ship, MV ERGE, which validated earlier reported remarkable energy efficiency onboard the single screw vessel. However, there is no recorded current studies in the public domain on application of a GRS on a twin screw vessel which are mostly Naval ships.

Naval ships have been reported to contribute a significant amount of CO₂ emission. These ships are currently said to emit more carbon emissions when compared with ships in the commercial sector. The emission of carbon by the naval ships could be mainly due to nature and operational characteristics and system design redundancy such as speed needed for naval operations (Giachetti & Moore, 2024). Additionally, Naval ships do not fall into the category of commercial shipping. Thus, the need to look into studying the application of a modified ESDs such as GRS for application onboard twin screw Naval ships so as for enhanced manoeuvrability, low propeller noise operation as well as energy efficiency.

This study will look into CFD computation of a bare hull for resistance validation and open water propeller simulation validation of a twin screw Naval ship. The geometry to be used is a DTMB 5415 Aleigh Burk class, United State Navy combatant and propose a suitable GRS for the vessel.

Several studies have been conducted since Sadotomo Kuribayishi, the chairman of Kuribayishi Steam co. proposed a hollow rectangular-type rudder to Japanese National Maritime Institute. Initial trials of the GRS, positioned the rudder aft of the propeller, yielded unsatisfactory results. However, subsequent design refinements involving optimization of both blade geometry and spatial orientation relative to the propeller demonstrated markedly improved performance characteristics (Professional Mariner Staff, 2020).

A recent hydrodynamic investigation conducted by Wärtsilä Netherlands B.V.'s R&D department in a study by (Opdam, 2024) examined the working principles of GRS as a unique ESD. The study addressed existing gaps in understanding the system's energy-saving mechanisms by focusing on hydrodynamic interactions between the GRS, propeller and ship's hull. The study evaluated two propeller rotation modeling techniques: Multiple Reference Frame (MRF) and Dynamic moving mesh approach for their suitability in GRS-propeller simulations.

2. Research Method

The research method in the study is the use of CFD code to validate experimental results for bare hull resistance and open water propeller simulation of a DTMB 5415. The CFD code used in the study is ANSYS fluent. The governing equations used for the analysis are Reynold's Average Navier-Stock equations (RANS).

$$\rho \frac{DV}{Dt} = -\nabla p + \rho g + \mu \nabla^2 V \quad (1)$$

$$\frac{\partial \rho}{\partial t} + \nabla(\rho u) = 0 \quad (2)$$

2.1 Bare Hull Resistance Validation

The study first conducted the bare hull resistance validation with available data of a DTMB 5415 experimental value. it is worthy to state that evaluating the total resistance in the study is analogous to a ship experiencing

resistance in the water which translate into thrust required by the propeller. See figure 2, the bare hull of the vessel.



Figure 2 Geometry of a DTMB 5415 bare hull form. Source: simman2014.dk

See the main particulars of DTMB 5415 at table 1.

Table 1: The Main Particulars for a DTMB 5415

Description (Principal)	Symbol	Unit	Value
Length between perpendiculars	LBP	m	142.037
Draft	T	m	6.179
Wetted surface area	A_{ws}	m^2	2976.7
Wetted plan area	A_{wp}	m^2	1987.2
Block Coefficient	-	-	0.506
Longitudinal center of gravity	LCG	m	71.6
Vertical center of gravity	VCG	m	1.39
Roll radius of gyration	K_{xx}	-	0.40 B
Pitch radius of gyration	K_{yy}	-	0.25 LBP
Yaw radius of gyration	K_{zz}	-	0.25 LBP
Speed	U	kn	{18;30}
Froude number	Fr	-	{0.28;0.41}
Water density	ρ	kg/m^3	998.5
Kinematic viscosity	ν	m^2/s	$1.09 \cdot 10^{-6}$
Gravity acceleration	g	m/s^2	9.803
Description (Model)			
Scale Ratio	-	-	24.832
Length	L	m	5.72
Draft	T	m	0.248
Wet Surface Area	A_{ws}	m^2	4.861

To compute the bare hull resistance, a computational domain was created using volume of fluid concept in ANSYS fluent design modeller. The details of the computational domain for the resistance validation is at Table 2.

Table 2: Computational domain Dimensions

Boundary conditions	Selected
Inlet: Front of the bow (Upstream)	1.0 LOA
Outlet: From the stern (Downstream)	3.0 LOA
Depth of air zone (Top)	0.5 LOA
Depth of water zone (Bottom)	1.0 LOA
Symmetry: Width of both zones (Transverse)	1.0 LOA

The meshing of the fluid domain in the model was conducted using ANSYS Meshing 2024R1 with CFD as the physics preference and Fluent as the solver. The element size was 0.25m. Inflation layers was 10 layers' maximum at 1.2 growth rate and 0.272 transition ratio. The meshing of the fluid domain was conducted for the analysis to properly represent the actual body, subsequently, mesh refinement was made on the face of Hull. The refinements are as shown in the table below:

Table 3: Mesh Refinement Details

Details	Coarse	Medium	Fine
Body sizing (m)	0.25	0.25	0.25
Face sizing (m)	0.02	0.0132	0.00975
Number of nodes	396458	626859	950054
Number of elements	1244557	1769618	2507861

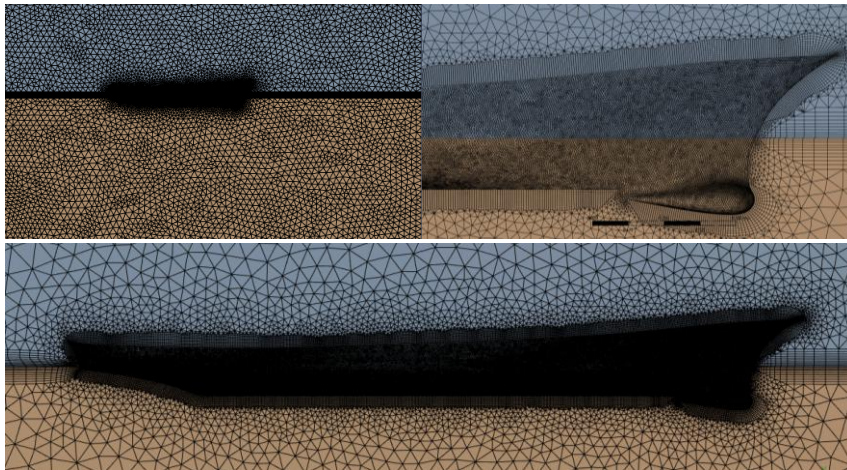


Fig.3. Meshing of bare hull and fluid domain.

The computation was conducted at different speed from the inlet. The result of the simulation is shown in Figure 5 showing.

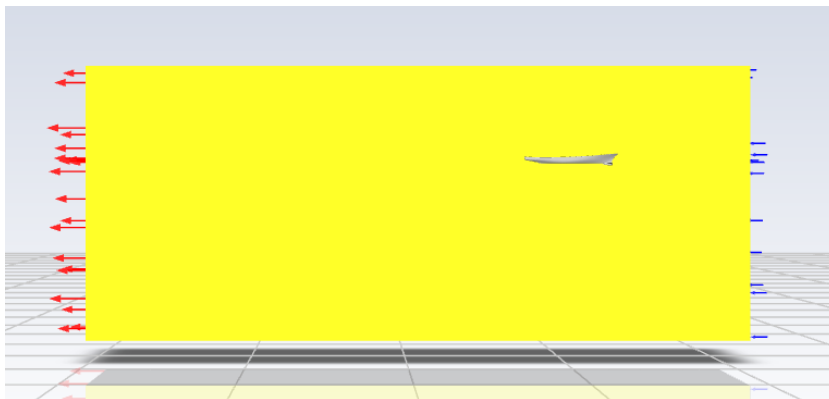


Fig. 4. Solution Domain showing inlet, outlet, hull and symmetry.

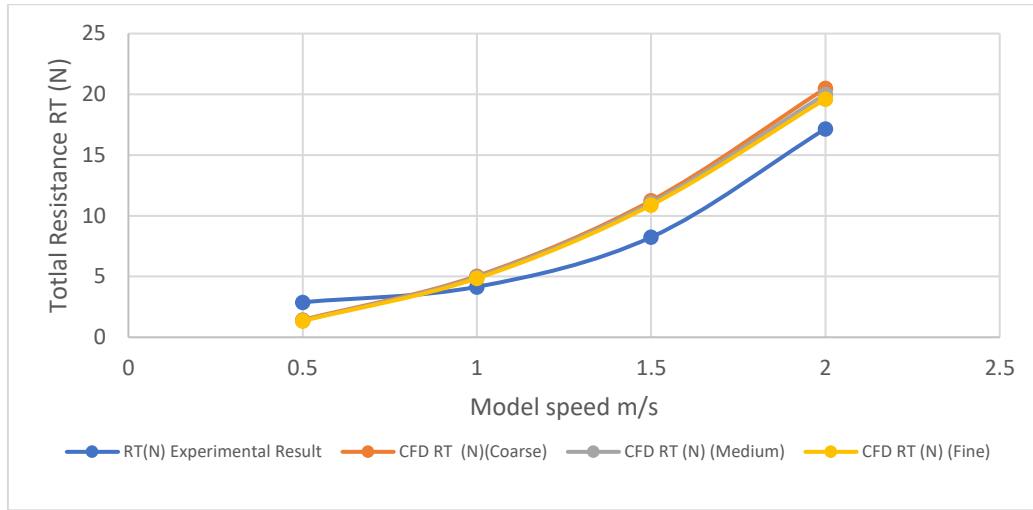


Fig. 5. Resistance curve for DTMB 5415 with different grid size at different speed.

After the resistance validation was completed, the next aspect of the study is open water propeller simulation.

2.2 Open Water Propeller Simulation

Open water propeller simulation is the simulation of ship propeller to determine its hydrodynamics characteristics in a homogeneous velocity field simulated with CFD codes [42] (Machfudin et al., 2023). The codes are used to compute the open-water hydrodynamic characteristics of a ship propeller. The major characteristics are the thrust coefficient K_T , torque coefficient K_Q and Efficiency η which are dimensionless. The equations to compute the J , K_T , K_Q and η are in equations 3,4,5 and 6 respectively. These coefficients are dependent on the advance coefficient J for a propeller that has constant pitch ratio P/D .

$$J = \frac{V_A}{nD}, \quad (3)$$

$$K_T = \frac{T}{\rho n^2 D^4}, \quad (4)$$

$$K_Q = \frac{Q}{\rho n^2 D^5}, \quad (5)$$

$$\eta_0 = \frac{J K_T}{2\pi K_Q}, \quad (6)$$

Where V_A is the velocity of advance, n is the speed in revolution per second (rps) D is the diameter of the propeller, T is the trust force of the propeller, ρ is the density, Q is the Torque of the propeller and η_0 is the propeller efficiency.

The DTMB propeller, P4508 was prepared using Rhinoceros software for the simulation. Table 4 shows the details of the propeller. Thereafter, the computational domain for the pre-processing was also prepared in Rhinoceros. For ease of computation and saving computational time, a section of the propeller with only one propeller blade was used in the computation. This is shown in figure 7.

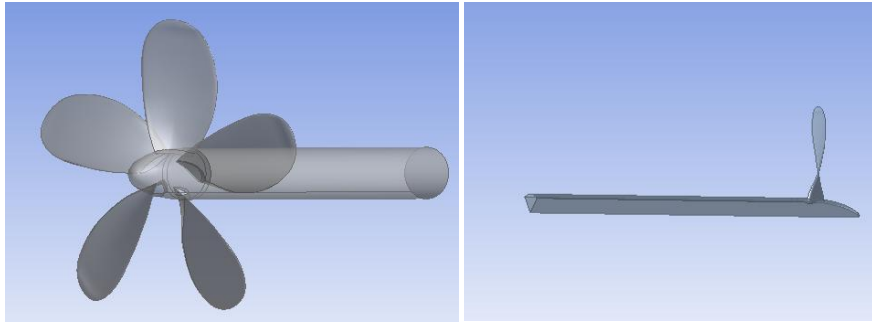


Fig.6. DTMB 5415 P4508 propeller and a Section of the propeller and shaft for pre-processing

Table 4: Details of DTMB 5415 Propeller

Description	Symbol	Model	Full
Scale		24.83	1
No. of blades		5	5
Diameter	D (m)	0.248	6.15
Pitch to Diameter ratio	P/D (0.7R)	0.870	0.870
Expanded Area Ratio	A_E/A_0	0.58	0.58
Rotation	-	Right	Right

The domain of the section of the propeller as seen in figure 7 was created using Boolean operation in ANSYS fluent design modeller. The length of the shaft is 5m and placed close the inlet. Details of the computational domain for pre-processing is in table 5.

Table 5: Computational domain for pre-processing DTMB 5415 P4508 propeller

Boundary conditions	Boundary
Inlet: Front of propeller blade	5m from propeller blade
Outlet: From propeller blade	3.5D
Top from propeller shaft	2D
Periodic boundary	72° from periodic boundary

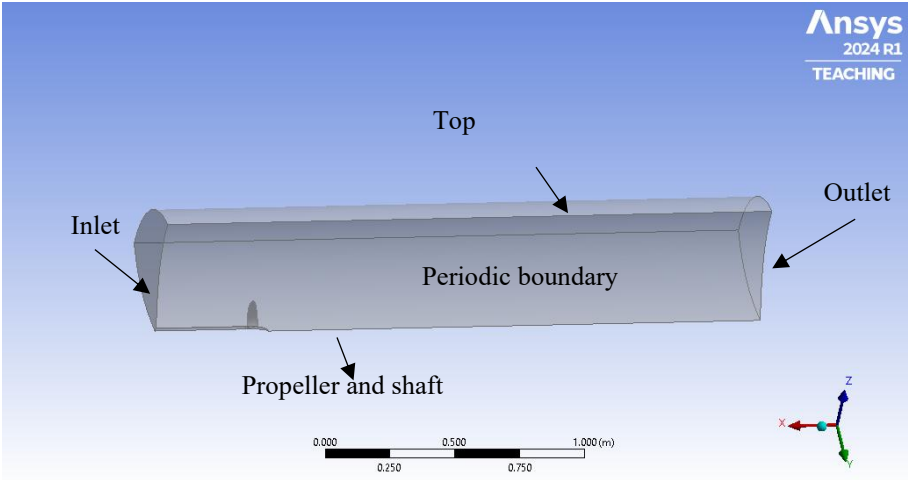


Fig.7. Computational domain of the propeller open water simulation

The meshing of the computational domain was done will poly-hex core elements. See meshing of a single blade propeller in figure 8. Details of the mesh refinement is at table 6.

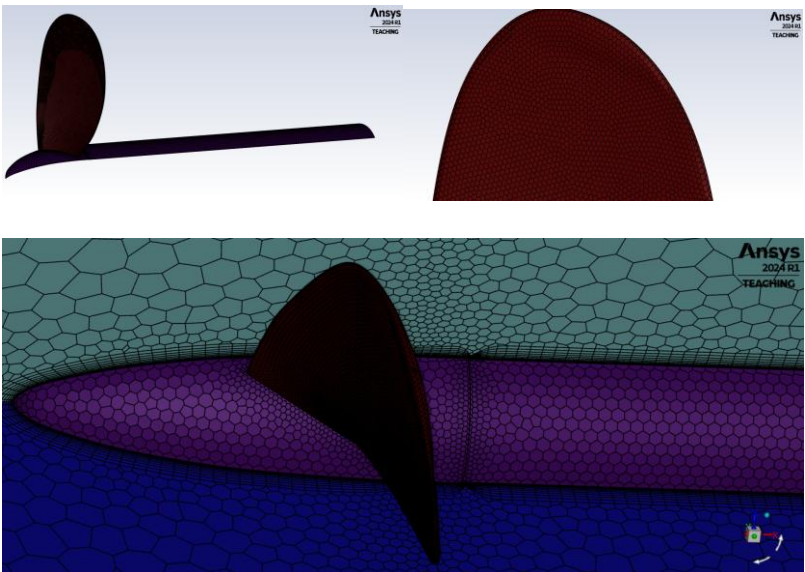


Fig 8: Meshing of the single propeller blade.

Table 6: Details of refinement

Details	Coarse	Medium	Fine
Body sizing (m)	0.001	0.001	0.001
Face sizing (m) (propeller blade)	0.0001	0.0007	0.0005
Face sizing (m) (propeller shaft)	0.0025	0.0025	0.0025
Number of nodes	740291	1260718	2308594
Number of elements	281824	485203	903194

The result of the computation for the open water propeller simulation using a single blade in the computational domain for a 0.248m diameter propeller at a constant speed of 20 rps is as shown in table 5. Grid convergence study was conducted by refining the coarse size of the mesh using a grid conversion ratio in accordance with ITTC recommendation as shown in Table 5.9 for both open water propeller simulation and bare hull resistance simulation. The grid spacing in both simulations agreed with the ITTC recommendation as the result showed monotonic convergence. $0 < Ri < 1$. With the value of 0.18 as the value of Ri .

Table 5.10: Grid Convergence Index. (Grid convergence ratio (r_G) = $\sqrt{2}$)

J	Vel (m/s)	Exp Data (N)	Total Trust Densities	(N) of Different Mesh	Mesh	ϵ_{32}	ϵ_{21}	R_i
			Fine Mesh (0.0005) 903,194 Elements	Medium Mesh (0.0007) 485,203 Elements	Coarse (0.001) 281,824 Elements			
0.5	2.48	347.63	336.95	329.55	285.55	43.45	7.95	0.18

Table 7: Result of CFD Open Water Propeller Computation Result

J	KT(CFD)	10KQ (CFD)	Efficiency (CFD)	KT(Exp)	10KQ(Exp)	Efficiency(Exp)
0.2	0.346625	0.48122	0.229186	0.35	0.48	0.232070868
0.3	0.301686	0.44949	0.320333	0.3	0.45	0.318268619
0.4	0.267688	0.41133	0.414133	0.27	0.41	0.419183059
0.5	0.223256	0.37984	0.467536	0.23	0.38	0.481590673
0.6	0.192449	0.3213	0.571748	0.19	0.32	0.566915977
0.7	0.149818	0.25732	0.648396	0.15	0.26	0.642657788
0.8	0.098458	0.18019	0.695427	0.1	0.18	0.707263597
0.9	0.049923	0.09867	0.724463	0.05	0.1	0.716104392
1	0.009883	0.03013	0.521787	0.01	0.003	0.530447698

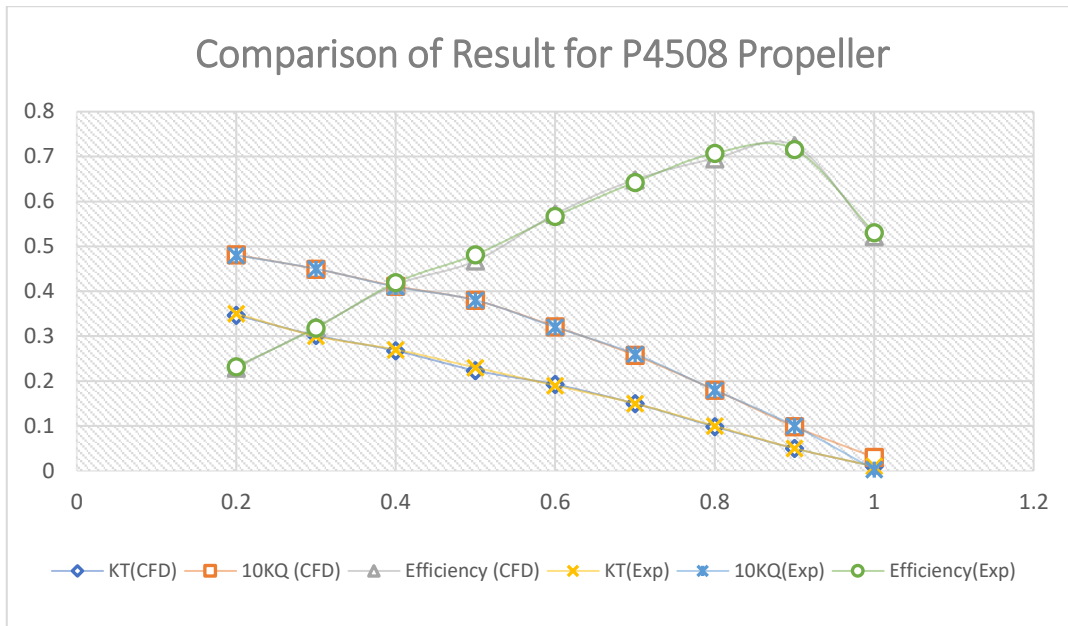


Fig.9. Open Water Performance result comparison between CFD and Experimental result.

3. Findings and Discussion

The analysis demonstrates that CFD analysis can be relied upon in computing hydrodynamics characters of a propeller and as well as bare hull resistance of a DTMB 5415 when compared with experimental data of a similar geometry of the same scale factor. This is evident from figure 5 which shows a curve showing bare hull resistance of the vessel at varying speed and from a result of open water experiment and a result from a CFD simulation. Also, figure 9 shows the Thrust coefficient, Torque coefficient and the Efficiency for an open water propeller performance experiment at different J number plotted together with a CFD simulation of a propeller with similar characteristics. Therefore, the study proposes a conceptual Gate rudder design for further CFD computation. It is worthy to note that the proposed gate rudder concept is different from those described in the literature as shown in figure 1. Since the current Gate rudder design has two rudder stud while the proposed gate rudder has only one stud there by reducing mechanical components onboard and possible increase in redundancy. The design parameters of the proposed gate rudder is in table 8 and the designed gate rudder is at figure 10.

Table 8: Design parameters for Gate rudder system

Description		Dimension (mm)
Vessel Draft		T
Propeller Diameter		D(0.85T)
Gate rudder height		1D
Gate Rudder Chord lengths	Lower cross section	0.3D
	Mid cross section	0.42D
	Top cross section	0.23D
	Top mid cross section	0.23D

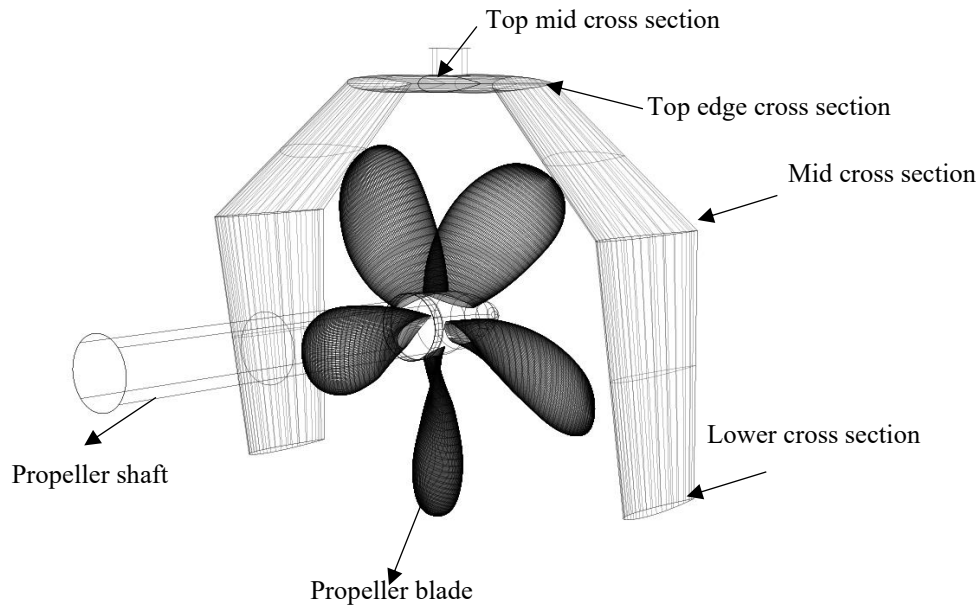


Fig.10. Gate Rudder Designed with one stud

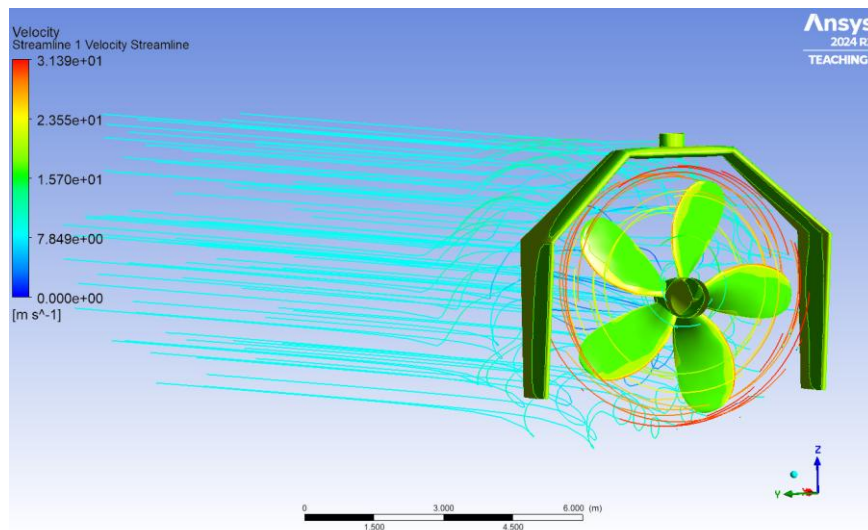


Fig.11. Gate Rudder Designed with one stud in a model during simulation showing velocity streamline

4. Conclusion

The application of a gate rudder to a twin screw naval ship is feasible. Despite that, the current studies on the subject of gate rudder systems were conducted on mostly cargo vessels, most notably is MV ERGE by the GATERS project which is not a high speed vessel when compared to a man of war. Thus, the need to investigate the viability of the reported energy efficiency of the energy saving device. Apparently, the study revealed that

CFD simulation could suffice for preliminary studies for a twin screw vessel with the validation of bare hull resistance and the open water propeller simulation of a DTMB 5415.

Declaration of Conflicting Interests

The authors declare no potential conflicts of interest with respect to the research, authorship and publication of this article.

Funding

The author received no financial support for the research, authorship and publication of this article.

References

- Opdam, W. (2024). Working principle of a gate rudder: A CFD analysis [Master's thesis]. [Eindhoven University of Technology, Netherlands]
- International Maritime Organization (IMO). (2023). 2023 IMO strategy on reduction of GHG emissions from ships. IMO.
- International Maritime Organization (IMO). (2015). Investigation of appropriate control measures (abatement technologies) to reduce black carbon emissions from international shipping. IMO.
- Sasaki, N., Kuribayashi, S., & Atlar, M. (2018). GATE RUDDER. Paper presented at INT-NAM2018, Istanbul, Turkey.
- Sasaki, N., Kuribayashi, S., & Miles, A. (2019). Full scale performance of Gate Rudder. Paper presented at RINA 2019, London, United Kingdom.
- Gürkan, A. Y., Ünal, U. O., Aktaş, B., & Atlar, M. (2023). An investigation into the gate rudder system design for propulsive performance using design of experiment method. *Ship Technology Research*, 71(2), 199–212. <https://doi.org/10.1080/09377255.2023.2248721>.
- Giachetti, R., Moore IV, H. (2024). Sustainable Systems: Measuring Carbon Emissions of Navy Ships. In: Salado, A., Valerdi, R., Steiner, R., Head, L. (eds) *The Proceedings of the 2024 Conference on Systems Engineering Research. CSER 2024. Conference on Systems Engineering Research Series*. Springer, Cham. https://doi.org/10.1007/978-3-031-62554-1_22
- Professional Mariner Staff. (2020, December 1). Advanced props, rudders provide new efficiencies below the waterline. Professional Mariner.
- Özsayan, S., Aydın, Ç., Köksal, Ç. S., Ünal, U. O., & Korkut, E. (2024). Assessment of the gate rudder system on the cavitation and underwater radiated noise characteristics of a containership. *OceanEngineering*, 313(Pt1), Article119391. <https://doi.org/10.1016/j.oceaneng.2024.119391>.
- Machfudin, A., Dinariyana, A. A. B., & Sari, D. P. (2023). VERIFICATION AND VALIDATION OF OPEN WATER TEST OF B4-65 B-SERIES PROPELLER MODEL. *Journal of Applied Engineering Science*, 21(4), 1052-1061. Article 1145. <https://doi.org/10.5937/jaes0-43599>.