MARINE PROPELLER DESIGN USING CFD TOOLS

Goutam Kumar Saha[1,a)] Md. Hayatul Islam Maruf[2,b)] and Md. Rakibul Hasan[2,c)]

[1]Department of Naval Architecture and Marine Engineering, BUET, Dhaka
[2]Department of Naval Architecture and Marine Engineering, Sonargaon University, Dhaka
a) gksaha2007@gmail.com
b) ih_maruf@yahoo.com
c) rakibname@gmail.com

Abstract. *A marine propeller generates adequate thrust to propel a vessel at some design speed. So, propeller is the core to optimum performance on a ship. Considerations are made to match the engine's power and shaft speed, as well as the size of the vessel and the ship's operating speed, with an appropriately designed propeller. As marine propeller has complicated geometries, the flow around the propeller is complicated. In generally, the performance characteristics of a propeller is determined and analyze by experiments like open water and self-propulsion model test which are costly at the initial stage of the design. Numerical analysis using Computational Fluid Dynamics simulations could be an important alternative on this case. This study presents the investigation of marine propeller hydrodynamic performance and parameters through Computational Fluid Dynamic analysis. In this paper, a B-series propeller model is developed with respect to some design constraints such as ship speed, vessel draft etc. and analyzed the performance using CFD tools. In this analysis, we consider Realizable $k-\omega$ Turbulence Model & Multiple Reference Frame Model. Results found that all thrust coefficient (K_T) and torque coefficient 10 (K_Q) decreases with the increasing advance coefficient (J). The efficiency of propeller performance had also consistently showed characteristic trend of nonlinear increases to a peak an optimum value before decreasing drastically with increasing J value. The numerical results obtained from CFD Tool are compared with theoretical available publish data.*

Keywords: *Propeller, CFD, ANSYS, B-Series Propeller*

INTRODUCTION

Marine propulsion is the mechanism or system used to generate thrust to move a ship or boat across water. When we consider about ship designs and design improvements, the propulsive efficiency is one of the most important issues. The marine propellers are necessary to be very efficient. The propulsive efficiency for ship designs is mandatory to be predicted and determined by model tests and full-scale observations in pre-design stage. Furthermore, there are many approaches and analysis have been done by researchers and naval architects in order to achieve the increasingly propulsive efficiency by comparing and investigating the ship hulls, propellers, rudders and energy saving devices. High marine fuel costs and low freight rates are causing operators to seek ways to boost ship efficiency.

Advanced ship propulsion solutions are one way to achieve considerable improvements in this regard but require highly detailed concept, design and construction processes. Model tests of self-propulsion were conventionally the only way to determine and evaluate the characteristics of hull-propeller interaction, the powering performance and the propulsion parameters of the ship. With the rapid advances in the field of computational fluid dynamics (CFD), numerical simulations of ship self-propulsion have recently gained increasing attention. Due to the increased capabilities of numerical flow simulations, it has become possible to make Computational Fluid Dynamics (CFD) analyses of self-propulsion calculations of ships with propellers**.** Computational fluid dynamics (CFD) provides ways of simulating flow fields around different geometries using numerical methods and established algorithms. During recent years, considerable progress in the field of computer science has contributed to the decrease of computational costs of CFD simulations, making it more accessible for practical applications. Nowadays, the role of CFD methods is increasing in most fluid dynamic's applications including the process of ship propeller design. Simulating the aforementioned experiments provides the opportunity to obtain desired results by analyzing calculated flow characteristics. It can be a practical way of obtaining valid results at relatively low costs and in reasonable time in relation to the real experiments. Since the self-propulsion test simulation is still quite expensive and time demanding, the common practice is to simulate only the open water test and to use its results for the determination of self-propulsion characteristics. It can be done by considering established interaction factors, which account for the interaction between the hull resistance and open water characteristics of the propeller. The difference between approaches lies in the level of simplification of the actual phenomenon as a trade-off between accuracy and CPU efficiency. The following two approaches, multiple frame of reference (MRF) for four bladed propellers with symmetric wall boundary condition and Steady simulation with rotating frame for simulating flow around a propeller are used and compared with them.

For the purpose of this research work steady state simulations with MRF model for a four bladed and complete propeller geometry were performed for different operating points of the propeller model. Thrust and Torque with respect to the advance ratio are obtained. As stated earlier, self-propulsion test simulation is quite expensive and time demanding. When modeling full interaction between the ship and the propeller, the main problem is the large difference between the time scales of the propeller and hull flow.

Different researchers are works on marine propellers based on CFD and FEA.

GOVERNING EQUATION

A propeller is a rotating fan like structure which is used to propel the ship by using the power generated and transmitted by the main engine of the ship. The transmitted power is converted from rotational motion to generate a thrust which imparts momentum to the water, resulting in a force that acts on the ship and pushes it forward. The simplest way to analyze propeller performance is to measure the thrust (T) produced by the propeller and the torque (Q) used to drive the propeller. Efficiency can be defined as the ratio of the acquired power in thrust production to the power used to drive the propeller. In the marine context, thrust and torque are usually given as non-dimensional measures as a function of a non-dimensional speed called the advance coefficient. The definitions of thrust- and torque coefficients, KT and KQ, and the advance coefficient, J, are given as

$$\boldsymbol{K_T = \frac{T}{\rho n^2 D^4}}$$

$$\boldsymbol{K_Q = \frac{Q}{\rho n^2 D^5}}$$

$$\boldsymbol{J = \frac{V_A}{nD}}$$

Above, propeller diameter is denoted as D, n is the number of propeller revolutions in one second, ρ is the water density and VA is the advance speed, for example the speed of a ship. The coefficients are found by applying dimensional analysis and assuming that free surfaces have no effect on the propeller performance. Under such conditions the coefficients are theoretically identical for all geometrically similar blade forms. The definition of efficiency can be written as

$$\boldsymbol{\eta = \frac{T.V_A}{2\pi n Q} = \frac{K_T}{K_Q}\frac{J}{2\pi}} \quad \text{.........} \quad (1)$$

The thrust and torque coefficients (K_T and K_Q) being similar for all cases with an identical blade shape is not entirely true. There are several factors that affect propeller performance even with a constant advance coefficient J.

Ship flows are governed by the three basic conservation laws for mass, momentum and energy, collectively referred to as the Navier-Stokes equations. The first law is mass conservation law-continuity equation which states that the rate of change of mass in an infinitesimally small control volume equals the rate of mass flux through its bounding surface.

Mass conservation – Continuity equation:

$$\boldsymbol{\frac{\partial \rho}{\delta t} + \nabla.(\rho V) = 0} \quad \text{...........} \quad (2)$$

where

∇ is the differential operator (∂/∂x, ∂/∂y, ∂/∂z)

ρ is water density (kg/m^3)

V = (u, v, w), fluid velocity (m/s)

Mass conservation – Continuity equation:

$$\boldsymbol{\frac{\partial(\rho u)}{\partial t} + \nabla.(\rho u V) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x} \quad (3)$$

$$\boldsymbol{\frac{\partial(\rho v)}{\partial t} + \nabla.(\rho v V) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y} \quad (4)$$

$$\boldsymbol{\frac{\partial(\rho w)}{\partial t} + \nabla.(\rho w V) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z} \quad (5)$$

Where, P is pressure, τ_{ij} is viscous stresses and f_i is body force.

Energy conservation – Energy equation:

$$\frac{\partial}{\partial x}\left[\rho(e + \frac{V^2}{2})\right] + \nabla.\left[\rho(e + \frac{V^2}{2})V\right] = pq + \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right) - \frac{\partial(up)}{\partial x} - \frac{\partial(vp)}{\partial y} - \frac{\partial(wp)}{\partial z} + \frac{\partial(u\tau_{xx})}{\partial x} \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z} + \frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v\tau_{yy})}{\partial y} \frac{\partial(v\tau_{zy})}{\partial z} + \frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} + \frac{\partial(w\tau_{zz})}{\partial z} + \rho f.V \quad \text{.....} \quad (6)$$

MODEL PROPELLER

For the CFD analysis, the first step is to design a 3D model of propeller. By using the basic data of B-Series propeller, a 3D propeller model is developed for analysis and is shown in Fig. 1. Particulars and geometric properties of the model propeller are shown in Table 1 and Table 2.

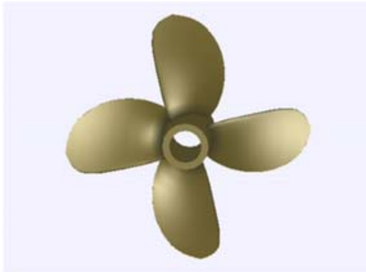

Figure 1: 3D Model Propeller

Table 1: Particulars of four bladed propeller

Particulars	*Dimensions*
Delivered Power, P_D	578.66 kW
BAR	0.55
Pitch Diameter ratio	0.69
Propeller Pitch	1.2486 m
Propeller Diameter	1.6029 m
Rake of GL aft	0^o
Skew Angle	17.6^o

Table 2: geometric properties of propeller

$^R/_R$	A	B	C	T (M)	R	MID C-GL	C/D
0.2	0.226	0.128	0.366	0.058	0.1603	-0.0429	0.2285
0.3	0.254	0.145	0.415	0.051	0.2404	-0.0469	0.2588
0.4	0.271	0.158	0.452	0.045	0.3206	-0.0456	0.2819
0.5	0.278	0.166	0.474	0.038	0.4007	-0.0408	0.2959
0.6	0.270	0.187	0.482	0.032	0.4809	-0.0294	0.3007
0.7	0.248	0.209	0.473	0.025	0.5610	-0.0113	0.2948
0.8	0.201	0.208	0.434	0.018	0.6412	0.0161	0.2709
0.9	0.122	0.174	0.349	0.012	0.7213	0.0520	0.2175
0.925	0.092	0.154	0.307	0.010	0.7414	0.0617	0.1915
0.95	0.051	0.123	0.246	0.008	0.7614	0.0720	0.1532
0.975	-0.007	0.076	0.152	0.006	0.7814	0.0834	0.0951
1.0	-0.093	0.005	0.010	0.005	0.8015	0.0974	0.0060

Where,

a = distance between leading edge and generator line at r.

b = distance between leading edge and location of maximum thickness.

c = chord length of blade section at radius r.

t = maximum blade section thickness at radius r.

PROPELLER PERFORMANCE ANALYSIS USING ANSYS FLUENT

ANSYS Fluent is a state-of-the-art computer program for modeling fluid flow, heat transfer, and chemical reactions in complex geometries. ANSYS Fluent provides complete mesh flexibility, including the ability to solve your flow problems using unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/ quadrilateral, 3D tetrahedral/ hexahedral/ pyramid/ wedge/ polyhedral, and mixed (hybrid) meshes. ANSYS Fluent also enables us to refine or coarsen our mesh based on the flow solution.

It is mentioned that generated mesh can be read into ANSYS Fluent, or, for 3D geometries mesh can be, create using the meshing mode of Fluent. All remaining operations are performed within the solution mode of Fluent, including setting boundary conditions, defining fluid properties, executing the solution, refining the mesh, and post processing and viewing the results.

After determination of the important features of the problem that want to solve, the basic procedural steps are shown below have followed.

1. Define the modeling goals.
2. Create the model geometry and mesh.
3. Set up the solver and physical models.
4. Compute and monitor the solution.
5. Examine and save the results.
6. Consider revisions to the numerical or physical model parameters.

Defining Model Geometry& Meshing

At the beginning of the process, the propeller geometry is imported into Ansys Design Modeler that was created in iges format and domain is generated as shown in Fig. 2. The Cartesian coordinates system is used, where x, y and z. The origin was located at the center of the hub, and positive directions are upstream, starboard and downstream. The domain dimensions were shown in Fig. 3. The solution field was divided into global and sub-domain. The sub-domain frame simulates the propeller rotation and employs the Coriolis acceleration terms in the governing equations for the fluid. The global frame surrounds the sub-domain frame. The global frame is a circular cylinder with 2 m radius and length is 12 m. The distance between the sub-domain frame and inlet is 1.991 m, while it is 9.991 m for the outlet. The sub-domain is also cylinder with a diameter of 0.018 m and length 0.018 m. Details of meshing properties are shown in Table 3 to Table 7.

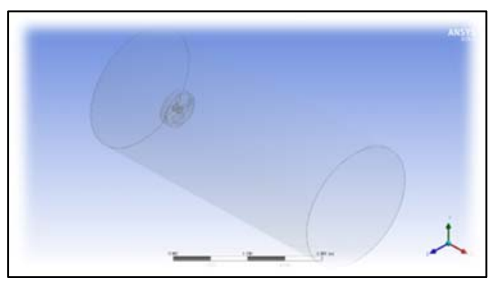

Figure 2: Propeller with Domain & Sub-Domain

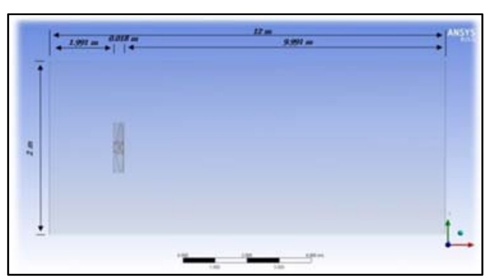

Figure 3: Dimensions of Domain & Sub-Domain

The generation of a suitable and fine grid is a crucial step in the simulation process. Promoting the negligible error in the procedure of the simulation and reducing the computational time for the program are the main issues for generating the mesh. There are two main methods to generate the mesh, one is to use the algebraic approach to make the grids, and the other one is to apply the partial differential equations to generate grids. The unstructured grids have the advantages of the lack of restriction on where points can be placed, in other words, it means it is possible to provide an automation of generation with a high degree. The mesh applied in these propeller models is the unstructured mesh which is also called the auto volume mesh, usually obtained after setting up all the parameters like inflations, the growth rate, the face sizing, the contact sizing, and the body sizing. This kind of mesh will reduce the cost of humanity but required a heavy work for the CPU of the computer. Therefore, reducing a very long computational time for each simulation is the basis issue of making mesh here. The models of the propellers are 3-D models, so the three-dimensional mesh is applied to the analysis of the 3-D models.

Table 3: Mesh Options & Its Position

Mesh Options	Position
Mapped Face Meshing	Outer Wall of Main Domain
Patch Conforming Method	Main Domain
Face Sizing	Propeller Blade
Inflation	Outer Wall of Main Domain

Table 4: Global meshing

Physics Preference	CFD
Solver Preference	Fluent
Element Order	Program Controlled
Size Function	Proximity & Curvature
Max Face Size	0.2 m
Mesh Defeaturing Size	2.74e-004 m
Growth Rate	1.15
Min Size	5.48e-004 m
Max Tet Size	0.3 m
Curvature Normal Angle	5°
Proximity Min Size	5.48e-004 m
Num Cells Across Gap	5
Boundary Box Diagonal	14.6620 m
Average Surface Area	11.2310 m^2
Minimum Edge Length	9.62e-003 m

Table 5: Inflation

Geometry	Main Domain
Boundary Scoping Method	Outer enclosure wall
Inflation Option	First Layer Thickness
First Layer Height	1.e-004 m
Maximum Layers	14
Growth Rate	1.2
Inflation Algorithm	Pre

Table 6: Statistics

Nodes	58,97,564
Elements	40,60,580

Table 7: Quality

Mesh Metric	Skewness
Min	1.3515e^{-006}
Max	0.84961
Average	0.2217
Standard Deviation	0.11467

Solver Setup

After all the alterations and adjustments are done in the meshing, the set up needs to be checked and modified before running the solutions.

This is done by the following ways

- Select the setup option from the work space.
- Select 3D for dimension
- Select Processing Option Serial

In the research work we will use, pressure-based solver type, absolute velocity function, transient time condition. For model properties, Set the viscous model is k-epsilon (2 equation), Realizable k-epsilon model and scalable wall function treatment near wall.

For materials properties, as the propeller is submerged at sea water, we select water for fluid and as our propeller is solid, we consider aluminum as a solid material. The density of water is 1025kg/m^3

In cell zone conditions, Main Domain is the stationary domains and Sub Domain is a rotating domain due to the blades part of the propeller located in it. Analysis type is a transient mode here in. And initial time in the automatic with value option is chosen, started with time at zero second. Details of cell zones conditions explained Table 8.

When the fluid flow goes through Main Domain and reaches the interface 1, which is in contact with the rotating propeller, a growing velocity will be applied to the flow. Accelerated flow goes across the interface 2 to pass through Main Domain whereas the mass flow at outlet is obtained. In the stationary domain, boundary conditions like the inflation layers and contacting sizing are imposed to provide an appropriate stationary environment. Details of boundary conditions are explained in Table 9.

Table 8: Cell Zone Conditions

Sub-Domain	Motion	Frame Motion
	Relative to Cell Zone	Absolute
	Rotation-Axis Origin	0,0,0
	Rotation-Axis Direction	1,0,0
	Rotational Velocity	-400
Main Domain	Motion	Stationary

Table 9: Boundary Conditions

Velocity Inlet	Reference Frame	Absolute
	Velocity Magnitude	Diff. Velocity Corresponding to J
	Coordinate System	(X, Y, Z) = (1,0,0)
Pressure Outlet	Backflow Reference Frame	Absolute
	Gauge Pressure	0 Pascal
	Backflow Direction	Normal to Boundary
Outer enclosure wall	Wall Motion	Stationary
	Shear Condition	No Slip
Propeller Blade	Wall Motion	Stationary
	Shear Condition	No Slip

Solution Method

The SIMPLE (Semi Implicit Pressure Linked Equations) scheme is used in the transient flow analyses. An implicit scheme is used when evaluating over time steps in the transient analysis. The second order upwind scheme is used for the Pressure, Momentum, Turbulent Kinetic Energy and Turbulent Dissipation Rate. The first order scheme is less accurate than the second order, but the stability is better.

And Finally, for starting the simulation we set the time step size for 0.003s for MRF. This value has to be matched with the rotating speed, when the sub domain is rotating, it should not skip any mesh element. In other words, the step size has to be smaller than mesh element, so all elements will be taken account in calculation. Otherwise it will be less precision in the calculation. More time step size and more iteration value increase, more time will be necessary for calculation.

RESULTS AND DISCUSSION

The continuous information has been contained in the exact solution of the differential equation describing the physical model would have been replaced with discrete values, in which, all grid nodes would have interactive properties between them and the neighbor nodes. A finite volume technique has been used in FLUENT package to convert the governing equations to algebraic equations that can be solved numerically. The convergence can be monitored dynamically by checking residuals. The residuals must be kept on decreasing from the start to end of the iterations, in this study the scaled residuals decrease to 10^{-5} for all equations. The numerical results have been obtained using about 1000 iterations to obtain a suitable level of solution convergence.

A direct comparison between numerical results and the experimental results has been carried out to validate the numerical model in the studied range. Validation has been carried out by calculating the thrust and torque coefficients at cases prescribed in experimental work results. The efficiency has been calculated and has been plotted to complete the total behavior of the marine propeller model. Eight different cases have been investigated as shown in Table 10:

Table 10: Values for Thrust and Torque Coefficient from CFD

Case	V_A	J	K_T	K_Q
1	*1.0686*	*0.1*	*0.297*	*0.034*
2	*2.1372*	*0.2*	*0.266*	*0.031*
3	*3.2058*	*0.3*	*0.232*	*0.028*
4	*4.2744*	*0.4*	*0.196*	*0.025*
5	*5.343*	*0.5*	*0.158*	*0.021*
6	*6.4116*	*0.6*	*0.120*	*0.018*
7	*7.4802*	*0.7*	*0.079*	*0.014*
8	*8.5488*	*0.8*	*0.033*	*0.010*

Pressure contour of propeller face and back are obtained at 400 rpm, considering 4.144 m/s inlet velocity and zero outlet gauge pressure. These contours are shown in Fig. 4 and Fig. 5. Here the maximum pressure is $1.028e^{+005}$ Pa. Also, the minimum pressure is $-8.216e^{+005}$ Pa.

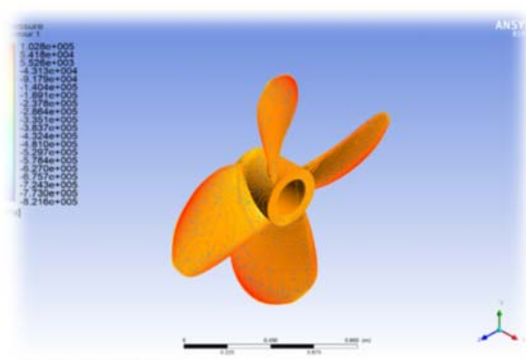

Figure 4: Pressure contour of Propeller Face

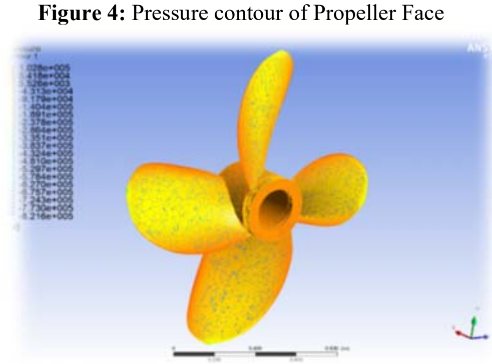

Figure 5: Pressure contour of Propeller Back

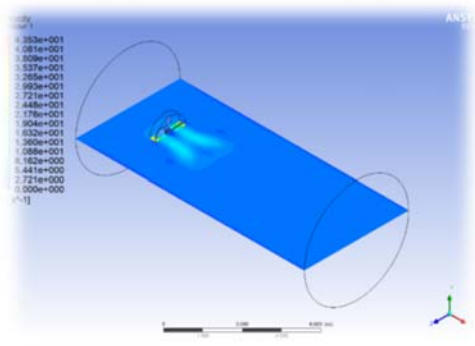

Figure 6: Velocity contour at stationary frame

The velocity contour at stationary frame is calculate at 4.144 ms^{-1} velocity, and 400 rpm and is shown in Fig. 6. At this boundary condition, the maximum velocity is $4.353e^{+001}$ $ms^{-1}$and the minimum velocity is 0.

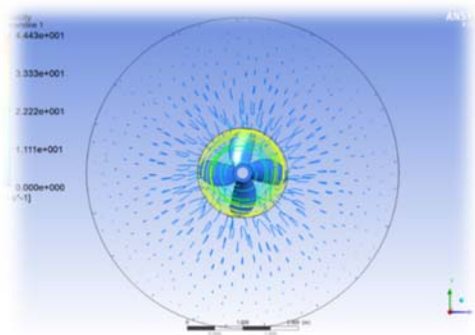

Figure 7: Velocity Streamline (sectional view)

The velocity streamlines are shown in Fig. 7 and 8. This are obtained for 400 rpm of propeller with inlet velocity 4.144 ms^{-}1. The maximum velocity of a streamline is found $4.443e^{+001}$ m/s and the minimum are 0.

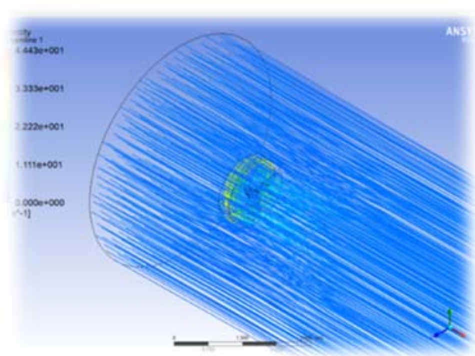

Figure 8: Velocity Streamline (Perspective view)

In order to validate the numerical results of CFD, the thrust coefficient and torque coefficient are also calculated (Table 11) using empirical method (Shiodu, A. et al. 2013) explained below:

$$K_Q=\sum_{n=1}^{47} C_n(J)^{S_n}\left(\frac{P}{D}\right)^{t_n}\left(\frac{A_E}{A_o}\right)^{u_n}(Z)^{v_n}$$

$$K_T=\sum_{n=1}^{39} C_n(J)^{S_n}\left(\frac{P}{D}\right)^{t_n}\left(\frac{A_E}{A_o}\right)^{u_n}(Z)^{v_n}$$

Table 11: Value for Thrust and Torque Coefficient from Empirical Formula

CASE	V_A	J	K_T	K_Q
1	*1.0686*	*0.1*	*0.262*	*0.029*
2	*2.1372*	*0.2*	*0.231*	*0.026*
3	*3.2058*	*0.3*	*0.197*	*0.023*
4	*4.2744*	*0.4*	*0.159*	*0.020*
5	*5.343*	*0.5*	*0.119*	*0.016*
6	*6.4116*	*0.6*	*0.076*	*0.012*
7	*7.4802*	*0.7*	*0.030*	*0.008*
8	*8.5488*	*0.8*	*-0.017*	*-0.017*

K_T, K_Q, and efficiency (η) values of both CFD results (Table -13) and empirical values (Table-14) are plotted with respect to advanced coefficient and is shown in Fig. 9. From these performance curves, the maximum efficiency of this propeller can be determined.

The maximum efficiency using empirical method is found to be 58.2% at a velocity 6.4116 ms^{-1} where in the case of CFD method, the maximum efficiency is about 64.6% at the velocity 7 ms^{-1}.

At our design speed 4.144 ms^{-1} corresponds to advance coefficient j equals to 0.4, the efficiency for empirical and CFD method are almost similar and is about 50%, but the empirical values are always slightly higher than the empirical ones.

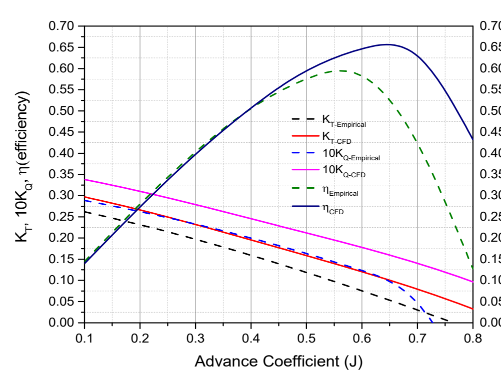

Figure 9: Performance Curve of Propeller

CONCLUSIONS

In this paper a design procedure of a marine propeller and its performance characteristics was discussed. It orders to design 3D the marine propeller PropCAD and Rhinoceros software were used. For the analysis of performances, Ansys Fluent was used. The CFD results were compared with empirical values and it is found that the CFD values always slightly higher than the empirical one. At the preliminary design stages, this method is useful to predict the performances of the propeller. In this work mesh convergence criteria were not carried out though it is required mesh convergence for CFD analysis.

REFERENCES

[1] Barru Harish, Kondapalli Siva Prasad, G. Uma Maheswara Rao, - "Static Analysis of 4-Blade Marine Propeller" -(Journal of Aerospace Engineering & Technology, Volume 5, Issue 2, 2015-ISSN: 2231-038X Online)

[2] Chang-Sup Lee1, Young-Dal Choi2, Byoung-Kwon Ahn1, Myoung-Sup Shin1 and Hyun-Gil Jang, "Performance optimization of marine propellers "(International Journal of Naval Architecture and Ocean Engineering, SNAK, 2010, Volume 2), Page 211-216

[3] Marian RISTEA, Adrian POPA, Dragos Ionut NEAGU- "CFD modeling of a 5 bladed propeller by using the RANSE approach"- "Mirceacel Batran" (Naval Academy Scientific Bulletin, Volume XVIII – 2015) – Issue 2.

[4] Mohamed M Helal, Tamer M Ahmed, Adel A Banawan and Mohamed AKotb- "Numerical prediction of the performance of marine propellers using computational fluid dynamics simulation with transition-sensitive turbulence model" - Proc IMechE Part M: J Engineering for the Maritime Environment 00(0). 2018

[5] Shiodu Anthony, Williams Ekwere, Ezenwa Ogbonnaya and Kuvie Ejabefio, "Design Procedure of 4-Bladed Propeller" - (West African Journal of Industrial and Academic Research Vol.8) - No.1 September 2013.

[6] Yanni Chang- "A comparative studyon the performances of different propeller designs", M.Sc. Thesis, University of South Eastern Norway, 2016.