

Hydrodynamic Analysis of Underwater Propeller

K.L.Satyavarma,^{a,*} and C. Neelima Devi,^b

^{a)} Department. of Mechanical Engineering, JNTUK-UCEV, Vizianagaram, INDIA

^{b)} Department. of Mechanical Engineering, JNTUK-UCEV, Vizianagaram, INDIA

*Corresponding author: satyavarma43@gmail.com

Paper History

Received: 1-October-2015

Received in revised form: 15-December-2015

Accepted: 30-December-2015

ABSTRACT

This paper presents a numerical study on hydrodynamic behaviour of underwater propeller for given performance conditions. Sound generated by a propeller is critical in underwater detection, and is often related to the survivability of the vessels especially for military purposes. The computations are conducted on the five bladed propeller. The RNG k- ϵ turbulence model with modified eddy viscosity coefficient is used for the computations, and the modified coefficient is related to the vapor and liquid densities in non-cavitating regions for simulating the non-cavitating flow. In this project, a suitable propeller will be identified for its strength in Non-cavitating condition, geometric model will be generated using the CATIA V5, R20, numerical analysis is carried out in ANSYS15 using FLUENT software, the propeller is studied for its hydrodynamic behavior for its Pressure and Velocity Contours, Thrust and Torque coefficients and comparing them with standard Theoretical formulae. The flow field is analyzed with finite volume method (FVM) Computed results are shown to be in good agreement with theoretical results.

KEY WORDS: Thrust and Torque coefficients; FVM.

NOMENCLATURE

K_T Thrust Coefficient
 K_Q Torque Coefficient

1.0 INTRODUCTION

Thrust is a reaction force described quantitatively by Newton's second and third laws. When a system expels or accelerates mass in one direction, the accelerated mass will cause a force of equal magnitude but opposite direction on that system. The force applied on a surface in a direction perpendicular or normal to the surface is called thrust. In mechanical engineering, force orthogonal to the main load (such as in parallel helical gears) is referred to as thrust [1]. Torque, moment or moment of force, is the tendency of a force to rotate an object about an axis, fulcrum, or pivot. Just as a force is a push or a pull, a torque can be thought of as a twist to an object. Mathematically, torque is defined as the cross product of the lever-arm distance vector and the force vector, which tends to produce rotation. Loosely speaking, torque is a measure of the turning force on an object such as a bolt or a flywheel. For example, pushing or pulling the handle of a wrench connected to a nut or bolt produces a torque (turning force) that loosens or tightens the nut or bolt [2, 3]. Ships and under water vehicles like submarines, torpedoes and submersibles etc., uses propeller as propulsion.

The blade geometry and its design are more complex involving many controlling parameters. The hydrodynamic analysis of such complex 3D blades with conventional formulas will give less accurate values. In such cases numerical analysis (Finite Element Analysis) gives comparable results with experimental values. Such complex analysis can be easily solved by finite element method techniques [4]. In the present case propeller consists of five blades. The diameter of propeller is 0.4 m and hub to propeller diameter is 0.389. In the present simulations for prediction of non-cavitating hydrodynamics of propeller is carried out at rotating speed of propeller at 780 rpm and the flow speed at 7.17 m/s using of Thrust and Torque coefficient equations coupled with RNG k- ϵ turbulence model computer code based on cell-centered finite volume method (FVM) on unstructured meshes for viscous flow field around propeller. These results are compared with theoretical formulae. [5] This paper addresses the flow behavior and normal force acting on a plate subject to oscillatory flow for KC numbers ranging from 1.4 to 105. For this purpose

3D CFD simulations were conducted on a U-shaped water tunnel configuration believe to be the same used .In their experimental investigation. With each flow oscillation vortices are shed from the tip of wall-mounted bilge keel plates installed at the middle of the water tunnel. The strength of such vortices is characterized by the KC number as first described .The KC number is calculated. The normal force on the plate can be characterized by drag and inertia components associated with coefficients C_d and C_m respectively. [6] The present thesis deals with modeling and analyzing the propeller blade of an underwater vehicle for their strength. A propeller is a complex geometry which requires high end modeling software. The solid model of propeller is developed in CATIAV5R20. Auto mesh is generated for the model using ANSYS Workbench. Hydro Dynamic analysis of aluminum alloy propeller is carried out in ANSYS- CFD FLUENT using SIMPLE for pressure and velocity coupling and Least Square Cell based for spatial discretization and second order up wind momentum and pressure equations first ordered time implicit scheme with time step 0.000254 and k- ϵ turbulence model are used. The Thrust and Torque coefficients are obtained are well validated with Theoretical formulae. The Theoretical formulae are referred from [7].

2.0 NUMERICAL INVISTIGATION

Self-propulsion simulations are time dependent due to propeller rotation. However, the body force propeller models often used in the simplified hull-propeller interaction analysis are usually incorporated with time-averaged flow fields and therefore steady flow approaches can be applied. In other applications of ship hydrodynamics, flow fields are inherently unsteady.

For spatial discretization, finite-volume method (FVM) with formally second-order accuracy was predominantly adopted. This seems to indicate that an increasing number of CFD practitioners in ship hydrodynamics prefer unstructured grids mainly due to ease of meshing and time-saving they offer. In these calculations turbulence effects were considered using turbulence models, as the k- ϵ RNG models, with the modification of the turbulent viscosity for multiphase flow. To model the flow close to the wall, the standard wall-function approach was used, and then the enhanced wall-function approach has been used to model the near-wall region (i.e., laminar sub layer, buffer region, and fully-turbulent outer region). For this model, numerical scheme used is segregated implicit solver. For the model discretization, the SIMPLE scheme was employed for pressure-velocity coupling, second-order upwind for the momentum equations, and first-order upwind for other transport equations (e.g., vapor transport and turbulence modeling equations [4].

The most used time discretization scheme is the first-order Euler implicit scheme. In cases where steady flow solutions are computed, the Euler implicit scheme is the natural choice for the unsteady solvers since time accuracy is not needed and a large time step is desirable for faster convergence. The hydrodynamic values such as thrust (K_t) and torque (K_q) coefficients and the other selected values were measured in this numerical research work.

2.1 Computational Mesh:

Mesh adaption/refinement/generation techniques that can adapt to

the shock front have been found to be a key ingredient in achieving accurate solutions for this kind of flow fields. Modeling, geometry, computational domains, boundary conditions, topology, meshing method and mesh size and turbulent method have significant effects on a fruitful numerical analysis and accuracy of simulation. Meshing strategy is divided in two divisions. Hybrid unstructured a mesh means that the tetrahedral elements for flow fluid fields, while structured mesh means that the hexahedral mesh is totally used for meshing on the solid surfaces. In contrast, the results of simulations with structured mesh elements usually have more accuracy than tetrahedral mesh elements results. CFD simulation data were verified with existing tests results.

Unstructured mesh elements production is almost automatic while hexahedral mesh elements generation is not automatic and should be generated manually. On the other hand, for flow field meshing, sometimes, the geometry is not compatible to use the hexahedral mesh elements, so unstructured mesh elements have better results and convergence of solution is nice [8]. Therefore, we used the hybrid unstructured mesh elements for rotational domain, in which we utilized the stationary and rotational domain for full scale propeller simulation for propeller with five blades. Auto mesh option is used in this project. Convergence is checked with element sizes 10 and 12.Close results are observed for element size 10 and 12.Element size 10 is used for mesh generation. After discretization number of elements in the domain are 7363445.The meshed figure of the propeller enclosed with its domain is shown in figure 1:

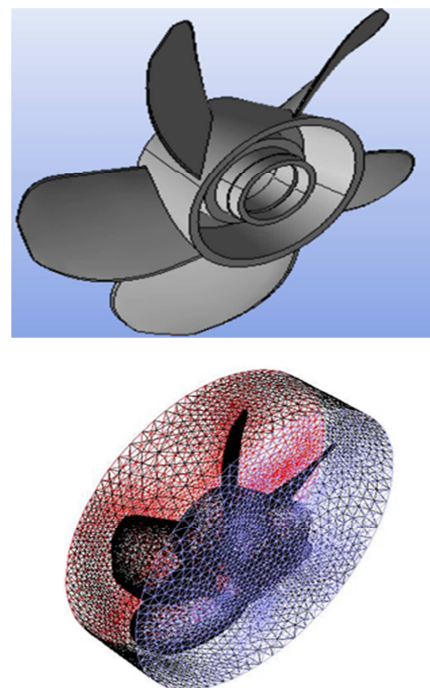


Figure 1: Geometry & meshed model of the propeller with its domain.

3.0 RESULTS AND DISCUSSIONS:

In the present case propeller consists of five blades. The diameter of propeller is 0.4 m and hub to propeller diameter is 0.389. In the present simulations for prediction of non-cavitating hydrodynamic behaviour of propeller is carried out at rotating speed of propeller at 780rpm and the flow speed at 7.08m/s these are shown in Table 1. The strength of the propeller is very much importance to warship designers and military strategists for many years. So in this case an attempt is made to prediction of hydrodynamic strength of propeller using of CFD Fluent software with k-e computer code based on cell-centred finite volume method (FVM) on unstructured meshes for viscous flow field around propeller and comparing these results with theoretical formulae.

Table 1: Principle particulars of propeller model.

Diameter of the Propeller	0.4m
EAR= A_E/A_0	0.58
No. of Blades	5
Hub ratio	0.389
Series	Naca

3.1 Analytical Procedure for Hydrodynamic Analysis in a Blade section by using CFD output i.e., Thrust, Torque:

The thrust and torque coefficients are none dimensionalized as follows

$$K_T = \frac{T}{\rho n^2 D^4} \quad (1)$$

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

where T is the thrust, Q is the torque, n is rps and D is diameter of the propeller.

Here we use the output, which is obtained from the computational dynamic analysis i.e., Thrust, Torque value.

As from CFD analysis we obtained Thrust as 2230.8372 N and Torque as 446.166 N-m.

Therefore, Thrust (T) = 2230.8372N

$$\text{Thrust coefficient } K_T = \frac{T}{\rho n^2 D^4}$$

$$= \frac{2230.8372}{2770 \times (13)^2 \times (0.4)^4}$$

$$K_T = 0.208$$

Torque (Q) = 446.166 N-m.

$$\text{Torque coefficient } K_Q = \frac{Q}{\rho n^2 D^5}$$

$$= \frac{446.166}{2770 \times 13^2 \times 0.4^5}$$

$$= 0.09181$$

3.2 Analytical Procedure for Hydrodynamic Analysis in a Blade section by following standard design formulae:

The following mathematical formulae are enclosed with reference [7]

$$\text{Brake power } P_B = 85HP \text{ or } 63.3844KW$$

$$\text{Ship speed } V_S = 7.08 \text{ m/s}$$

$$\text{RPM (n)} = 840$$

$$\text{Speed of Advance } V_A = V_S (1 - w) \quad (3)$$

$$V_A = 11.6981 \text{ knots}$$

$$\text{Shaft power } P_S = P_S \eta_s \quad (4)$$

$$= 83.3HP$$

$$\text{Loading constant } P_D = P_S \eta_S \quad (5)$$

$$= 81.634 HP$$

$$\text{Power coefficient } B_p = P_D^{0.5} n / V_A^{2.5} \quad (6)$$

$$= 15.0571$$

$$= 15$$

From the chart of type B series of 5-blades shown, the value of $B_p = 15.0$ is read. The point of intersection b/w the B_p line and optimum line (in red line) was traced to get $(P/D) = 0.91$, $\eta_0 = 0.662$ and $\delta_{opt} = 155$, Diameter of the propeller D= 0.4m.

Having determined the pitch, diameter and delivered horse power of the propeller, the thickness blade, the thickness blade, the blade area & hub (boss) diameter from the ratios stated for these in the type B series chart for 5 blades design are as follows:

Number of blades (Z) = 5

$$\text{Blade area ratio } \left(\frac{A_E}{A_0} \right) = 0.58$$

$$\text{Blade thickness ratio} = 0.112$$

$$\text{Blade Area (Disk area) } A_0 = \frac{\pi D^2}{4} \quad (7)$$

$$A_0 = 611.1427 \text{ ins}^2$$

Weight of all blades is equal to total thrust acting on the propeller

$$w = T = 1.982 B_{tf} \zeta Y R^3 \quad (8)$$

$$= 2824.2389 N$$

Now, we can use this obtained weight/ force as thrust value[7], Therefore, the thrust T = 2824.2389N.

$$\text{Thrust coefficient } K_T = \frac{T}{\rho n^2 D^4}$$

$$K_T = 0.2635$$

For finding of Torque value we have to use

Torque $Q = F \times d / 2$

$$Q = 564..8477 \text{ Nm}$$

$$\text{Torque coefficient } K_Q = \frac{Q}{\rho n^2 D^5}$$

$$= 0.09168$$

3.3 Pressure and Velocity Profiles:

From CFD Analysis we directly obtained Thrust and Torque, Pressure and Velocity contours by taking 1000 iterations in k-ε turbulence model by taking time step size equal to 0.000256. The convergence graph is shown in fig 2.

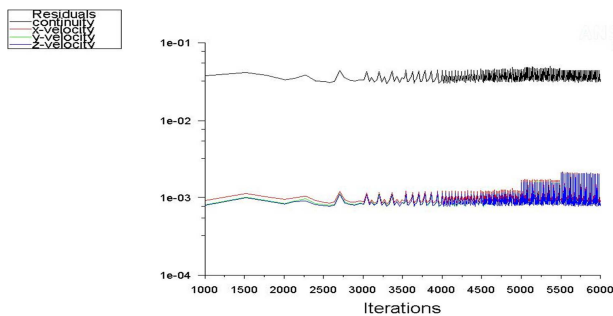


Figure 2: Convergence graph

The graph represents the convergence history of the propeller sound pressure levels.

The convergence criteria are considered as the difference between the values of the succeeding and preceding are in the range of 0.001.

The figures 3 & 4 represent the Contours of the Pressure and Velocities at various sections of the propeller.

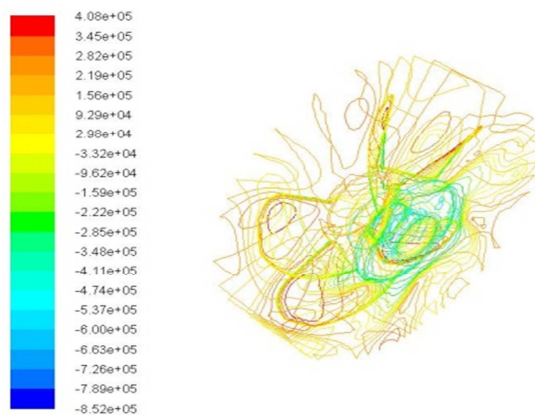


Figure 3: Contours of Static Pressures (Pa) representing inlet, outlet, outer, wall solid.

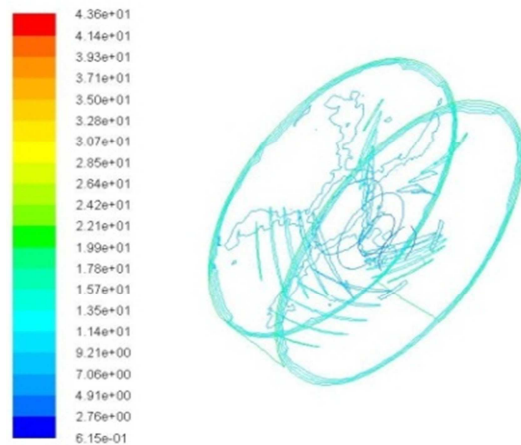


Figure 4: Contours of Velocity (m/s) of inlet, outlet, and wall solid.

4.0 CONCLUSION

The propeller is assumed to be operated at 780 rpm with forward velocity of 7.08m/s for prediction of thrust and torque coefficients. The convergence is obtained after completion of 1000 iterations for this k-ε turbulence model is used. The Thrust coefficient values obtained from both ways are as follows, 0.208 & 0.2635. The Torque coefficient values are obtained from both ways are 0.09181 & 0.09168. The Thrust and Torque coefficients obtained from both CFD and Formulae are quite acceptable and within the ranges.

5.0 ACKNOWLEDGEMENTS

The authors wish to express their sincere gratitude to Sri V. Rama Krishna, Scientist'D', NSTL, Visakhapatnam and Mrs. C. Neelima Devi, Assistant Prof., University College of Engineering, Vizianagaram, JNTUK.

6.0 REFERENCE

1. T.R. Kane and D.A. Levinson, "Dynamics, Theory and Applications", 1985, pp. 90–99.
2. Hendricks, Subramony, and Van Blerk, Chinappi "Physics for Engineering", page 148, Web link
3. Kleppner, Daniel; Kolenkow, Robert "An Introduction to Mechanics". McGraw-Hill. pp. 267–68.6/30/2014 *Torque - Wikipedia, the free encyclopaedia* (1973).
4. J. P. Abraham et al, "Turbulent and transitional modeling of drag on oceanographic measurement devices", *Trends in Naval Architecture Applications, Final report and Recommendations* to the 27th ITTC
5. Z. Boutanios, "CFD Analysis of a U-shaped Water Tunnel Bilge Keel Experiment", 15th Numerical Towing Tank Symposium 7-9 October 2012 Cortona/Italy, Volker

- Bertram, Emilio Campana (Ed.)
6. Pierre-Luc Delafin, François Deniset, Jacques-André Astolfi, "Prediction of hydrodynamic forces with and without transition model", 15th Numerical Towing Tank Symposium 7-9 October 2012 Cortona/Italy, Volker Bertram, Emilio Campana (Ed.)
 7. Ishiodu Anthony, Williams Ekwere, Ezenwa Ogbonnaya and Kuvie Ejabefio, "Design Procedure of 4-Bladed Propeller", Department of Marine Engineering, Marine Academy of Nigeria, Oron, Akwa Ibom State, Nigeria. *West African journal of Industrial and Academic Research* Vol.8 No.1 September 2013.
 8. Sakamoto, N., Carrica, P.M., Stern, F. "URANS simulations of static and dynamic maneuvering for surface combatant: part 1. Verification and validation for forces, moment, and hydrodynamic derivatives," *J. Mar. Sci. Technol.*, Vol. 17, pp. 422–445, 2012.