GLOBAL JOURNAL OF ENGINEERING, DESIGN & TECHNOLOGY
(**Published By:** Global Institute for Research & Education)
www.gifre.org

CFD ANALYSIS OF CONTROLLABLE PITCH PROPELLER USED IN MARINE VEHICLE

Aditya Kolakoti[1] ,T.V.K.Bhanuprakash[2] & H.N.Das[3]

[1]M.E in Marine Engineering And Mechanical Handling, Dept of Marine Engineering, Andhra University College of Engineering, Visakhapatnam-India.
[2] Professor Dept of Marine Engineering, College of Engineering, Andhra University, Vishakhapatnam-India.
[3] Scientist 'F', Hydrodynamic Research Wing, NSTL, Visakhapatnam-India.

Abstract

For moving ahead ship needs thrust, which is generated by rotating propeller behind its hull. A controllable pitch propeller can adjust incidence of incoming flow and can offer better control over the thrust generated by it.
Traditionally thrust and torque of a propeller, is predicted by model test. This takes more time, manpower, space and cost. In present time CFD is getting popular for hydrodynamic designs as it give relative saving in time, manpower, space and cost.

In order to simulate the flow around a rotating propeller and ship, a computational domain is created surrounding the propeller and ship. Boundary conditions are set to simulate the actual condition. Mesh is generated in the domain and finally flow solution is obtained.

The flow analysis of propeller behind the ship hull will be carried out in three stages.
1. CFD analysis of Bare Hull of a ship.
2.Open water analysis of propeller
3.Flow characteristics of propeller when attached behind the same ship hull.
FLUENT®, was used for CFD analysis and for modeling and meshing the packages used are CATIA – V5® and ICEM-CFD® respectively.

Keywords: Resistance, Propeller, Wake, Thrust Deduction Fraction, Hull Efficiency, Relative Rotative Efficiency, Quasi- propulsive efficiency, Self propulsion point. Computational fluid dynamics.

1. INTRODUCTION

Controllable pitch propellers are propellers in which the blades are separately mounted on the hub, each on an axis, and in which the pitch of the blades can be changed, and even reversed, while the propeller is running, by means of an internal mechanism in the hub.

Estimation of self propulsion point Wake, Hull efficiency, Relative rotating efficiency and Thrust Reduction factor were calculated using CFD.

2. ANALYSIS OF BARE HULL

An operational ship is considered for analysis the length of the ship is L.

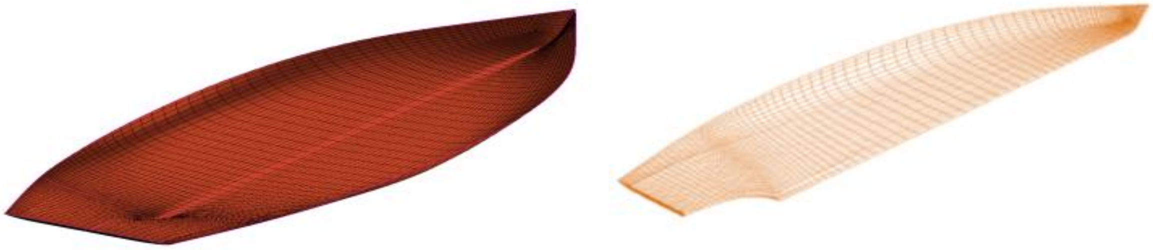


Fig.2.1 ***Geometric model and mesh over the hull***

Table.2.1 Ship Hull Details

Domain size	Rectangular domain of length 4Lm, Breadth 1Lm and Depth 1Lm

2.1 Grid Generation

The flow domain is required to be discretized to convert the partial differential equations into a series of algebraic equations. This process is called grid generation. Total number of elements was generated 81171 hexahedral cells.To generate the structural grid with hexahedral cells, the commercially available grid generation code ICEM CFD was used.

2.2 Boundary Conditions

The continuum was chosen as the fluid and the properties of water were assigned to it, which is stationary. The wall forming the draft part of the ship hull between interface and symmetry set as symmetry boundary condition .uniform velocities was at inlet. The far boundary (far field) was assumed to be an inviscid wall.

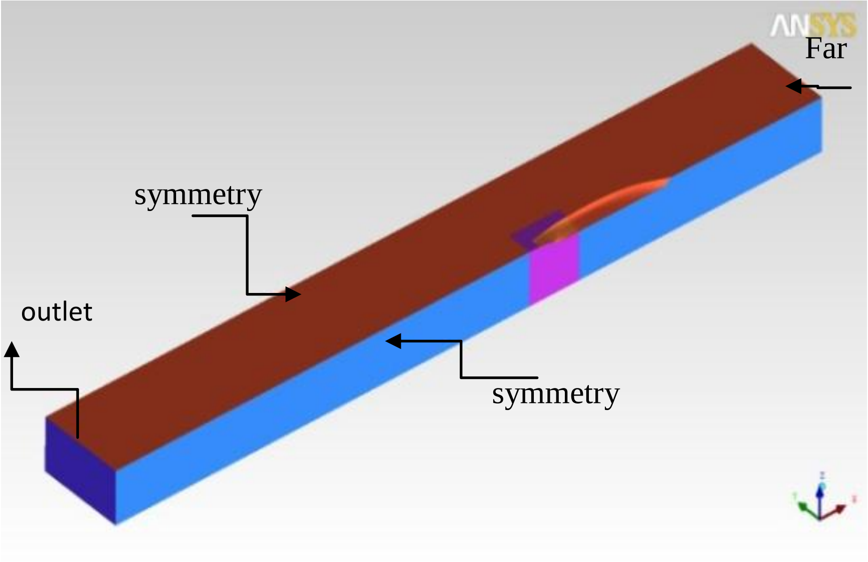


Fig.2.2 ***Boundary Conditions***

Table.2.2 Boundary Conditions

ZONE	TYPE
Inlet, Far	Velocity -inlet
Hull	Wall
Symmetry	Symmetry
Fluid	Water liquid
Outlet	Outflow

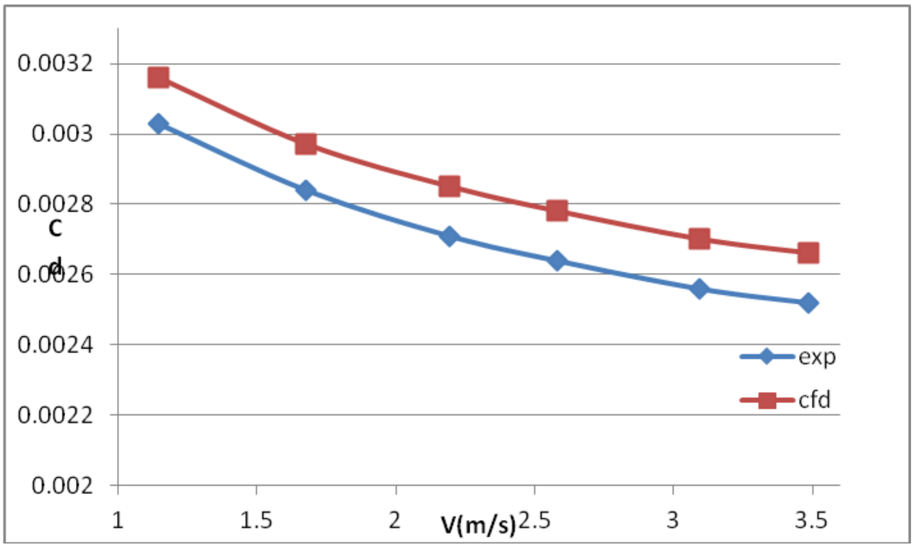
Solver	FLUENT
Pressure Link	SIMPLE
Descretisation Method	QUICK
Relaxation Parameter	0.1
Wall function	Standard
Turbulence model	K-ε

2.3 Results

CFD analysis for computations of resistance encountered by a ship is carried by considering with single phase flow and this is performed as six different velocities. FLUENT software were used for this purpose. The volume of displacement and wetted surface area for all the six configurations were taken as reference parameters to non-dimensionalise the resistance coefficients. Coefficient of drag includes viscous and pressure resistance (excluding wave resistance)

Table 2.3. Comparison Of CFD Results With Experimental Results for Ship Bare Hull

Froude no	Results from FLUENT	Experimental	
	Coefficient of Drag Resistance C_D	*Coefficient of Drag Resistance C_D*	*% difference*
0.12	**0.00316**	**0.00303**	**4.11**
0.18	**0.00297**	**0.00284**	**4.37**
0.24	**0.00285**	**0.00271**	**4.91**
0.28	**0.00278**	**0.00264**	**5.035**
0.34	**0.00270**	**0.00256**	**5.185**
0.38	**0.00266**	**0.00252**	**5.263**



Graph: 2.1 Variation of coefficient of Drag with speed

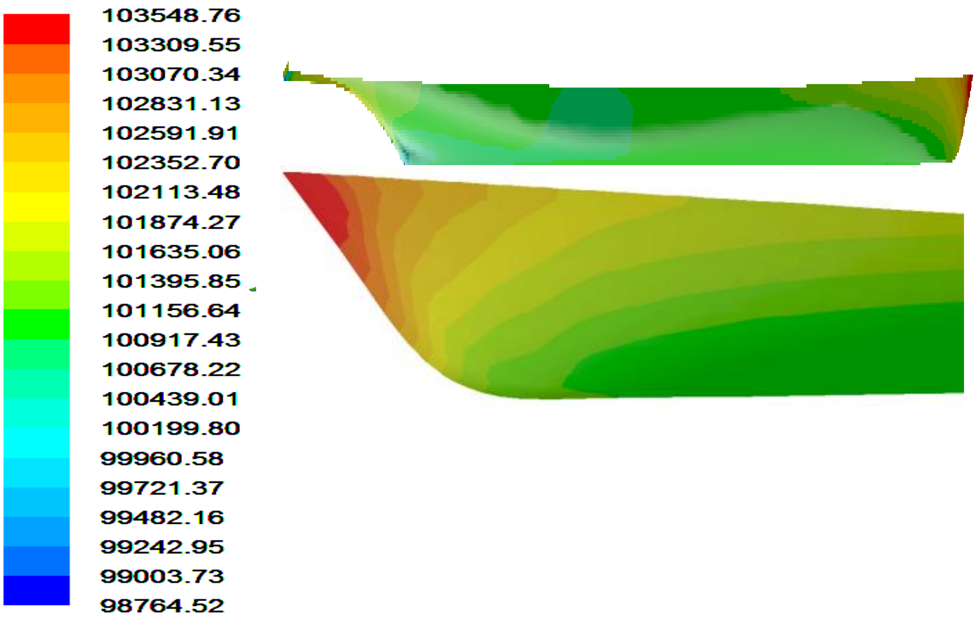


Fig:2.2 Pressure distribution over hull

3. OPEN WATER CHARACTERISTICS OF PROPELLER

An operational propeller of diameter D having 5five blades is considered for analysis.

3.1 Grid Generation

To generate the structural grid with hexahedral cells commercially available grid generation code ICEM CFD was used. Total number of elements was generated: 846001. The five blades are at a regular angular interval of 72 degrees. So modeling one angular sector with an extent of 72 degrees containing one propeller blade is sufficient to solve the entire flow domain. The effect of other blades was taken care by imposing periodic boundary condition on meridional planes at the two sides.

Table.3.1 Propeller Details

Propeller	Controllable pitch propeller
Principal Dimensions	Propeller Diameter, D
Domain size	Cylindrical domain of length 7D, dia 4D.
Number of blades	Z=5

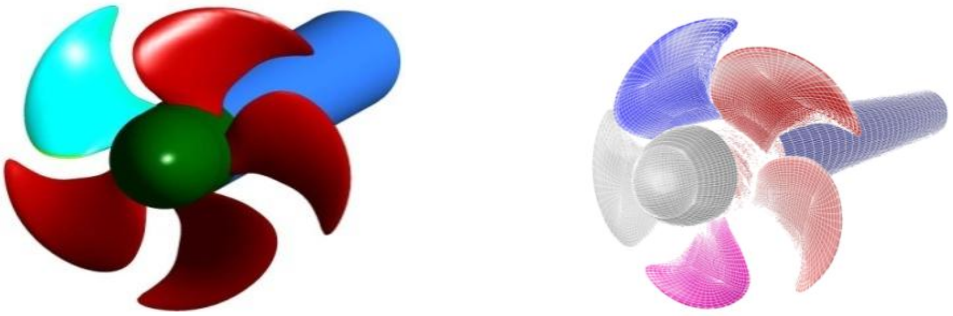


Fig.3.1 Geometric model and mesh over the propeller

3.2 Boundary Conditions

A moving reference frame is assigned to fluid with different rotational velocities at different J (advance ratio) values. The wall forming the propeller blade and hub were assigned a relative rotational velocity of zero with respect to adjacent cell zone. A uniform velocity was prescribed at inlet, at outlet outflow boundary condition was set and far boundary (far field) was also taken as velocity inlet and assigned a uniform velocity.

Table3.2 Propeller Flow Characteristics

Pressure Link	SIMPLE
Pressure	Standard
Turbulence model	Standard K-ε
Wall function	Standard
Solver	FLUENT

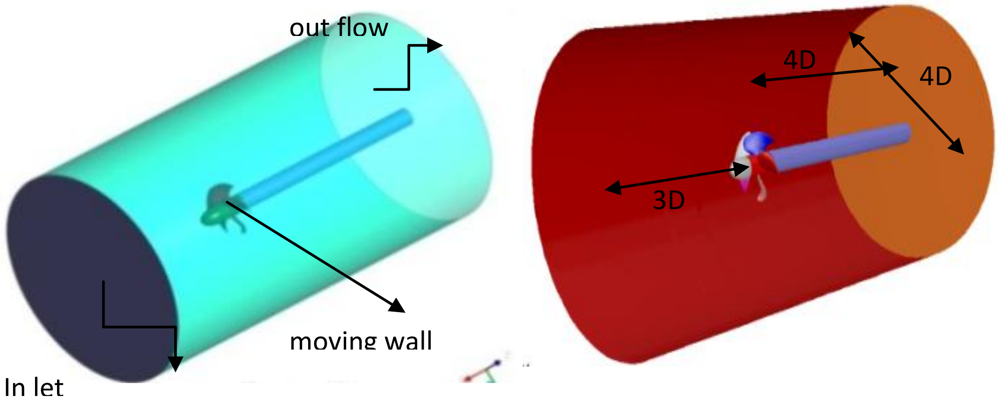


Fig.3.2 Boundary Conditions for Propeller

3.3 Results

A complete computational solution for the flow was obtained using Fluent software. The output of the software is the Thrust force T, and Torque (Q). From the results of the software, Thrust coefficient and Torque coefficient K_T and K_Q respectively were estimated. Table: 5.2 shows the results obtained from the software and those of the experiments.

Definition of Coefficients:

Advance Coefficient $$J = \frac{V_a}{nD}$$

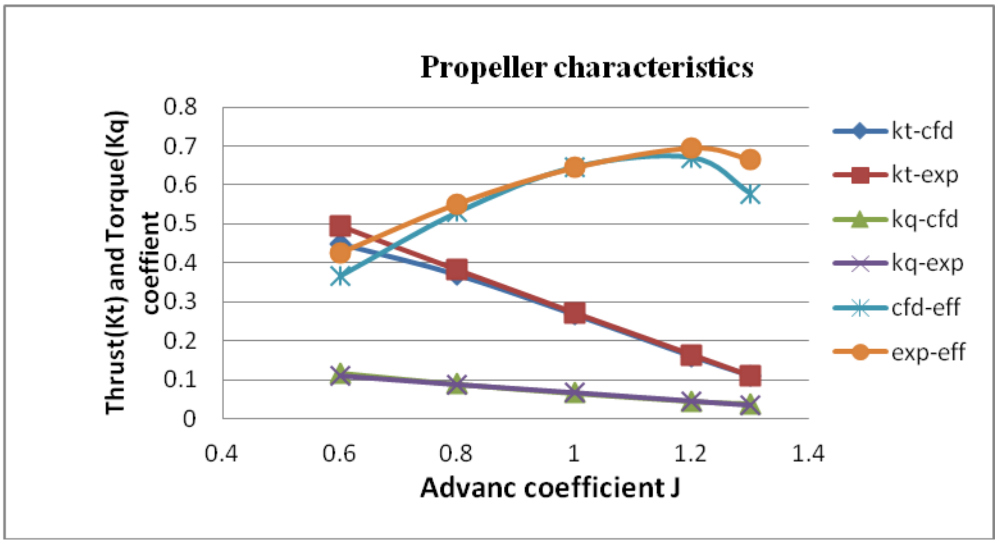
Thrust Coefficient $$K_T = \frac{T}{\rho.n^2.D^4}$$

Torque Coefficient $$K_Q = \frac{Q}{\rho.n^2.D^5}$$

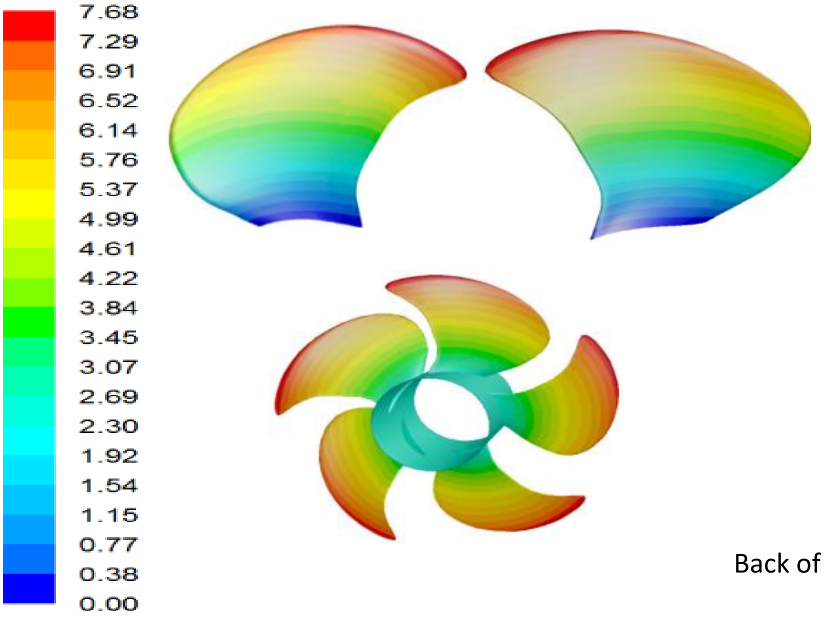
Efficiency: $$n = \frac{j}{2\pi} * \frac{K_T}{K_Q}$$

Table3.3 Computational Estimation Of K_T, K_Q & Efficiency For P/D=1.547

Advance Coefficient J	K_T		K_Q		Efficiency η	
	CFD	Exp	CFD	Exp	CFD	Exp
0.6	0.449	0.494	0.117	0.111	0.365	0.42
0.8	0.369	0.381	0.088	0.088	0.529	0.55
1.0	0.266	0.271	0.065	0.067	0.645	0.64
1.2	0.159	0.163	0.045	0.045	0.669	0.69
1.3	0.108	0.109	0.038	0.033	0.578	0.67



Graph: 3.1 Comparison of predicted K_T & K_Q with experimental data.

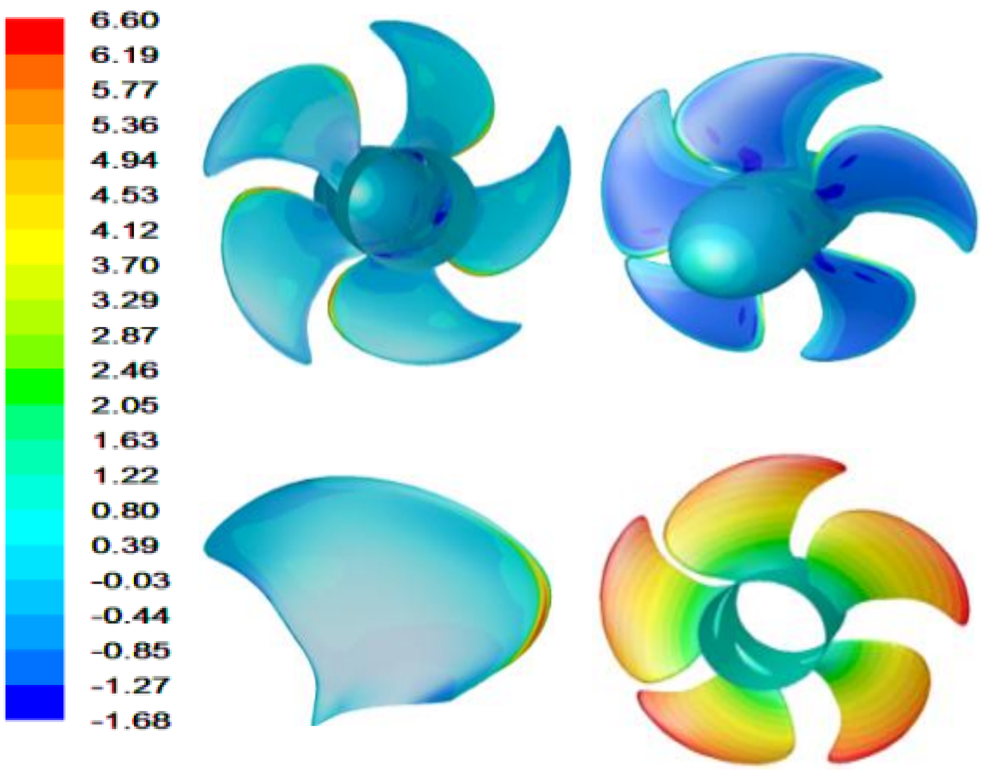


Back of the propeller

Fig: 3.3 Velocity distribution over the propeller

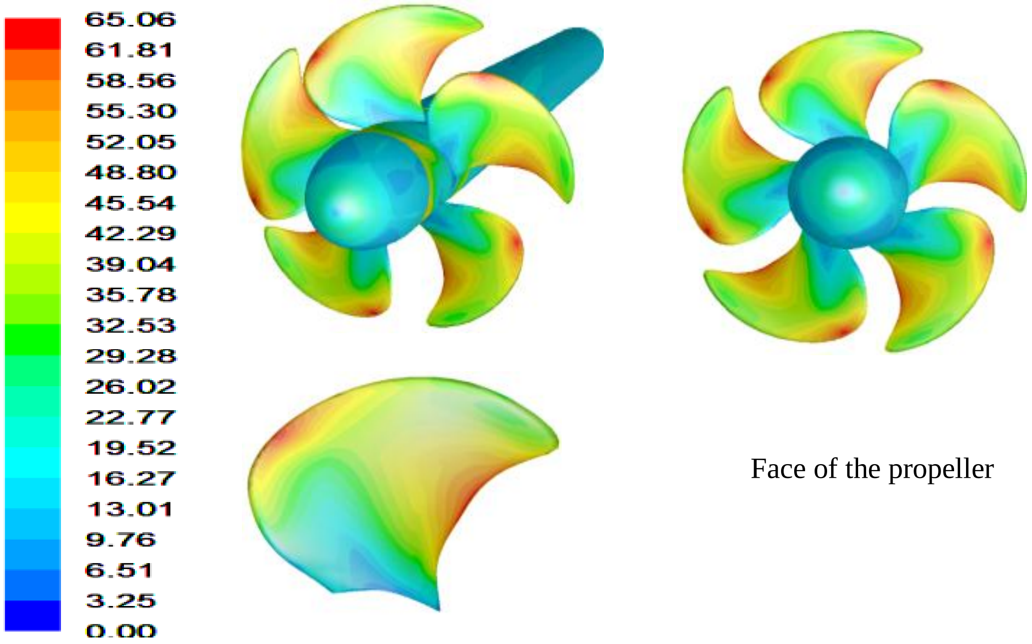
***Pressure Distribution Around The Blade-Surface*:**

Pressure distribution on the surface of blades is shown in Fig.3.4 The face and back is experiencing high pressure and low pressure respectively. This explains the development of thrust by propeller. It is evident that there is a concentration of high-pressure region near the leading edge of the propeller.



Face of the propeller Back of the propeller

Fig: 3.4 Contours of pressure coefficient for P/D=1.547 at j=1.0



Face of the propeller

Fig: 3.5 Contours of wall Y^+

4. PROPELLER ANALIZED AT BEHIND CONDITION

4.1 Wake

The difference between the ship speed *V* and the speed of advance may be called the *wake speed.* Froude expressed the wake speed as a fraction of the speed of advance calling this ratio the *wake fraction* , so that

$$w_f = \frac{v - v_a}{v_a}$$

=0.9387

4.2 Thrust Deduction Fraction

When a propeller produces thrust it accelerates the water flowing through the propeller disc and reduces the pressure in the flow field ahead of it. The increased velocity of water at the stern of ship and reduced pressure cause an increase in the resistance of the ship. If R_T is the total resistance of the ship at a given speed in the absence of the propeller and R_T' is the total resistance at the same speed when the propeller is producing a thrust T, then the increase in resistance due to the action of propeller is:

$$\delta R = R_T{}' - R_T$$

And $$T = R_T{}' = R_T + \delta R$$

Since this increase in resistance is an effect due to the propeller, it is convenient to put $\delta T = \delta R$ and to write:

$$T - \delta T = R_T$$

Where δT is the "thrust deduction". The ratio of thrust deduction to the thrust is called the thrust deduction fraction *t*. The thrust deduction is found to be

$$t = \frac{\delta T}{T}$$

=0.9454

4.3 Hull Efficiency

The work done in moving a ship at a speed *V* against a resistance R_T is proportional to the product R_T V or the effective power P_E . The work done by the propeller in delivering a thrust *T* at a speed of advance V_A is proportional to the product T V_A or the thrust power P_T. The ratio of the work done on the ship to that done by the screw is called the *hull efficiency*, so that

$$n_H = \frac{P_E}{P_T} = \frac{R_{TV}}{TV_A}$$

or

$$n_H = \frac{1-t}{1-w}$$ in Taylor notation

=0.89070

4.4 Relative Rotative Efficiency

The propeller in open water, with a uniform inflow velocity, at a speed of advance V_A has an *open water efficiency* given by

$$n_o = \frac{TV_A}{2\pi n Q_o}$$

=0.67

Where Q_0 is the torque measured in open water when the propeller is delivering thrust *T* at *n* revolutions. Behind the hull, at the same effective speed of advance V_A the thrust *T* and revolutions will be associated with some different torque *Q*, and the *efficiency behind the hull* will be

$$n_B = \frac{TV_A}{2\pi n Q}$$

=0.619

The ratio of behind to open efficiencies under these conditions is called the *relative rotative efficiency*, being given by

$$n_R = \frac{n_B}{n_o} = \frac{Q_o}{Q}$$

$$= \frac{0.619}{0.67}$$

= **0.923**

4.5 Quasi-Propulsive Efficiency

Quasi-propulsive efficiency equals hull efficiency times relative Rotative efficiency times open propeller efficiency.

$$n_D = n_H \, n_R n_o$$

$$= 0.89*0.93*0.66$$

= **0.54**

4.6 Self Propulsion Point

Self propulsion point (CFD) was estimated around 127N at 20.3rps.Self propulsion point (Experimental) was estimated at 125.91N at 18.13rps.

Table 4.1 Computational Estimation Of K_T, K_Q & Efficiency When The Propeller Behind The Hull

Advance Coefficient J	CFD		
	K_T	K_Q	Efficiency η
0.6	0.4555	0.1072	0.40
0.8	0.3308	0.0852	0.49
1.0	0.2024	0.0604	0.53
1.2	0.1569	0.0493	0.58
1.3	0.0091	0.0173	0.10

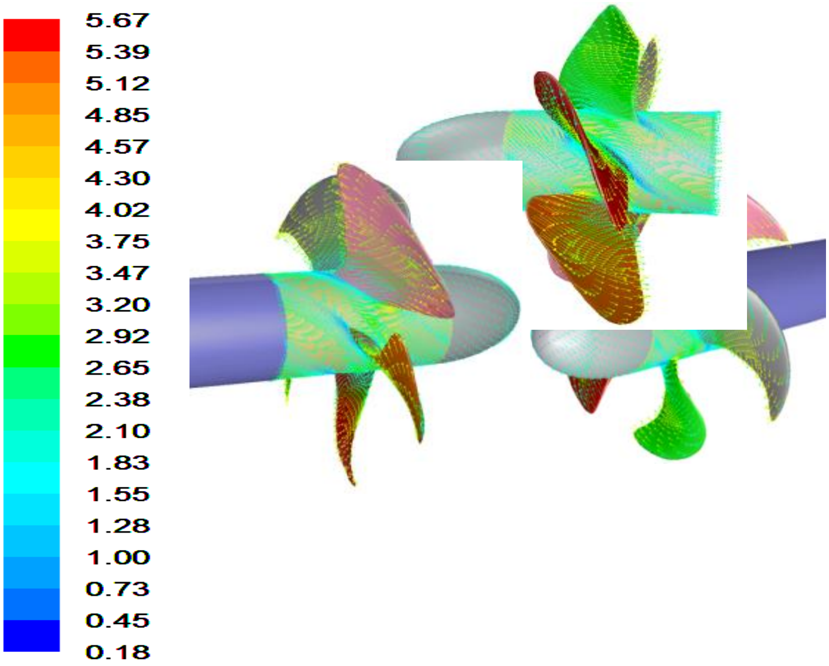


Fig: 4.1 Velocity magnitudes around the propeller

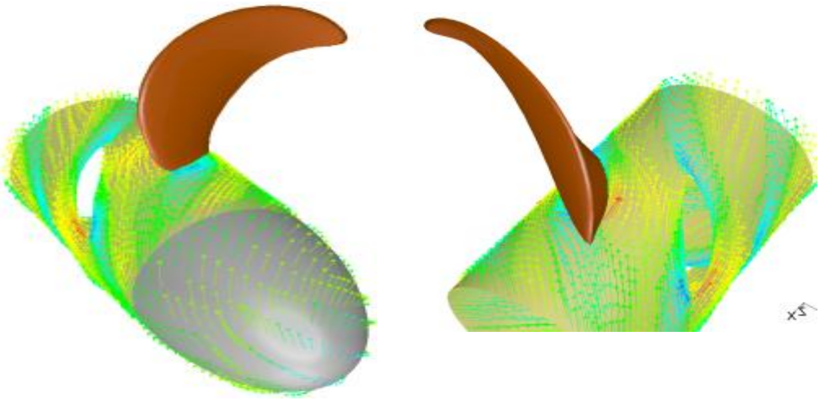


Fig: 4.2 Velocity Vectors at hub of the propeller

5. CONCLUSIONS

- CFD study was carried out for bare hull, propeller characteristics at open water and behind conditions.
- For behind condition, wake fraction, relative rotating efficiency and quasi-propulsive efficiency were calculated.
- Results were validated against experimental and shown good agreement
- The difference between CFD and experimental is around 4% to 6%. The difference in thrust and torque estimation is around 4% and 14% respectively in the operational regime of the propeller. At higher advance ratios where propeller thrust, torque and efficiency reduce considerably, the difference grows larger.
- The self propulsion point for the experimental and CFD results are compared and found very minute variation in

results which is acceptable

- CFD was used successfully for estimating propeller characteristics in a complicated flow region like hull-propeller interaction. Coefficients estimated here can readily be used for ship design which will be highly beneficial for designers.

6. ACKNOWLEDGMENTS

Authors express the sense of gratitude to **Shri S V Rangarajan, Distinguished Scientist**, Director (N.S.T.L), for giving permission to undertake this project work in Technical Directorate of Hydrodynamic Research.

7. REFERENCES

1. Anderson, J., 1995, Computational Fluid Dynamics, McGraw–Hill Inc., Newyork
2. Banawan, A. A. and Ahmed, Y., 2006, Use of computational fluid dynamics for the calculation of ship resistance and its variation with the ship hull form parameters, Marine Eng. and Naval Architecture Dept., Faculty of Eng., Alexandria University, Alexandria, Egypt.
3. Hoffman,K. A. and Chang, S. A., 2000, Computational Fluid Dynamics, Engineering Education System, Kansas.
4. Lewis, E. V., 1988, Principles of Naval Architecture, The Society of Naval Architects and Marine Engineers, Jersey city, NJ.
5. Quing-Lin, R.,2007, Modelling The Effects Of Ship Appendages On The Six-Degree Of Freedom Of Ship Motions, Hydrodynamics department, David Taylor motor basin, NSWCCD.
6. Sridhar.D, Bhanuprakash.T, and Das, H. N., 2010, Frictional Resistance Calculations on a Ship using CFD, International Journal of Computer applications, Visakhapatnam.
7. Versteeg, H. K., and Malalasekhara, W., 1995, An Introduction to Computational Fluid Dynamics, Longman Group Limited, England.