

Performance Analysis and Enhancement of Marine Propeller

Shreyash C. Godge

Bachelor of Engineering Student
Department of Mechanical Engineering
Shree L. R. Tiwari College of Engineering, Mira Road,
Thane, Maharashtra, India

Aditya M. Pandya

Bachelor of Engineering Student
Department of Mechanical Engineering
Shree L. R. Tiwari College of Engineering, Mira Road,
Thane, Maharashtra, India

Shashank P. Shet

Bachelor of Engineering Student
Department of Mechanical Engineering
Shree L. R. Tiwari College of Engineering, Mira Road,
Thane, Maharashtra, India

Dheeraj K. Amin

Bachelor of Engineering Student
Department of Mechanical Engineering
Shree L. R. Tiwari College of Engineering, Mira Road,
Thane, Maharashtra, India

Abstract—This analysis of a propeller is based off the coordinates of model KCD 32 (Emerson and Sinclair, 1967). Using SOLIDWORKS to recreate the geometry of a three-dimensional geometry, analysis was conducted. The study is completed using a computational program, Ansys FLUENT, and velocity, pressure distribution, torque is compared to experimental results. Reasonable results are produced such that the torque and efficiency trends will be in acceptable limits with respect to experimental data. The acquired results are used as input data to carry out stress analysis on propellers made of three composite materials namely carbon composite, alumina composite and polymer composite.

Keywords—HydroEnergy, Optimization, FSI interaction, Blade design, CFD analysis, Marine Propeller, Advance coefficient, Torque, Torque coefficient, Material, Convergence, etc.

I. INTRODUCTION

A propeller is a type of fan that transmits power by converting rotational motion into thrust. A pressure difference is produced between the forward and rear surfaces of the aero foil-shaped blade, and a fluid (such as air or water) is accelerated behind the blade.

Propeller dynamics, like those of aircraft wings, can be modelled by either or both Bernoulli's principle and Newton's third law.

A marine propeller of this type is sometimes colloquially known as a screw propeller or screw, however there is a different class of propellers known as cycloidal propellers – they are characterized by the higher propulsive efficiency averaging 0.72 compared to the screw propeller's average of 0.6 and the ability to throw thrust in any direction at any time. Their disadvantages are higher mechanical complexity and higher cost.

II. HISTORY OF PROPELLER

In 1835, two inventors in Britain, John Ericsson and Francis Pettit Smith, began working separately on the problem. Smith was first to take out a screw propeller patent on 31 May, while Ericsson, a gifted Swedish engineer then working in Britain, filed his patent six weeks later. Smith quickly built a small model boat to test his invention, which was demonstrated first on a pond at his Hendon farm, and

later at the Royal Adelaide Gallery of Practical Science in London, where it was seen by the Secretary of the Navy, Sir William Barrow. Having secured the patronage of a London banker named Wright, Smith then built a 30-foot, 6-horsepower canal boat of six tons burthen called the Francis Smith, which was fitted with a wooden propeller of his own design and demonstrated on the Paddington Canal from November 1836 to September 1837. By a fortuitous accident, the wooden propeller of two turns was damaged during a voyage in February 1837, and to Smith's surprise the broken propeller, which now consisted of only a single turn, doubled the boat's previous speed, from about four miles an hour to eight. Smith would subsequently file a revised patent in keeping with this accidental discovery.

III. OBJECTIVES

- To study different types of propeller.
- To perform extensive literature survey.
- To simulate experimental test in Emerson paper [1], numerically using commercial Software ANSYS Fluent 18.0
- To study effect of change in material with composite material, in overall efficiency of marine propeller.

IV. PROBLEM DEFINITION

A propeller is the most common propulsor on ships, imparting momentum to a fluid which causes a force to act on the ship. The ideal efficiency of any size propeller (free-tip) is that of an actuator disc in an ideal fluid. An actual marine propeller is made up of sections of helicoidal surfaces which act together 'screwing' through the water (hence the common reference to marine propellers as "screws"). Three, four, or five blades are most common in marine propellers, although designs which are intended to operate at reduced noise will have more blades. CFD study of propeller can be used study its different problem like propeller induced vibration, tip erosion, propeller cavitation, singing propeller, etc. A good

CFD model of propeller's working can be utilized to study cause of above problems and methods to prevent it

V. METHODOLOGY

A propeller is a type of fan that transmits power by converting rotational motion into thrust. In this project we have studied different types of propeller. After we have selected marine propeller for designing of 3D model.

The modelling of marine propeller was done in Solidworks 2015 software. By using this software time requires in producing the complex 3D model and risk involved in design and manufacturing process can be minimized. After modeling complex 3D model it was further imported to the CFD analysis.

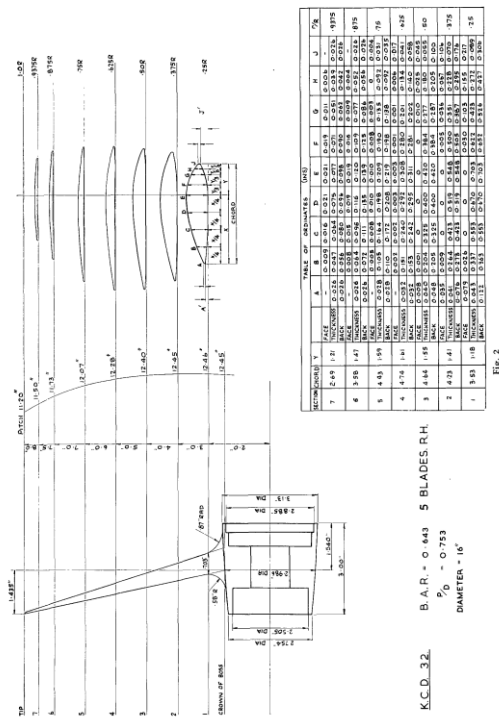
The overall system of marine propeller is analyzed using CFD analysis. CFD study of propeller can be used study its different problem like propeller induced vibration, tip erosion, propeller cavitation, singing propeller.

After selection we have design 3D model propeller of different shape and sizes using Solidworks software. This model was further used to develop CFD model to get effect of rotation of marine propeller at different advance coefficient values (J). The CFD data and different settings were used with reference to previous literature work used in research done on wind turbines.

The results obtained from CFD analysis was transfer to static structural analysis as a part of 1-way FSI simulation. The pressure over blade surface was mapped to surface of marine propeller blade for loading boundary condition and blade was fixed at shaft. This process was performed on 4 different blades of different materials. This was performed to achieve proposed objective of reducing weight of blade, without affecting working of marine propeller.

VI. MARINE PROPELLER SPECIFICATION

The coordinates of the blades were based on a 5- bladed right-hand propeller; model KCD 32 (Emerson and Sinclair, 1967). The coordinates are set up as shown in Figure 5.4.1 There are 7 sections (foils) at varying pitch and distance from the hub:



Propeller Cavitation: Systematic Series Tests on 5- and 6-Bladed Model Propellers 227

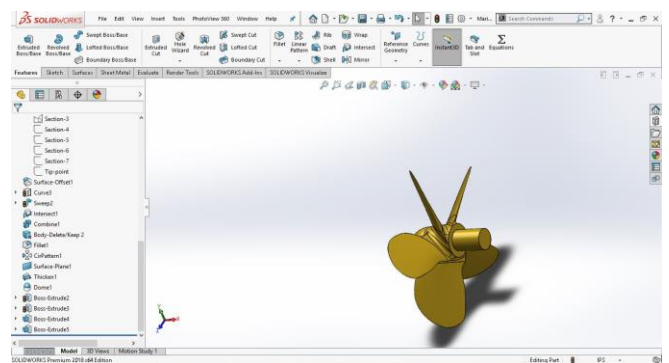


Figure 1 Final 3D model of marine propeller Blade in Solidworks 2015

VII. COMPUTATIONAL FLUID DYNAMICS ANALYSIS

A. Mathematical Model: Governing Equations

The governing equations are the continuity and Navier-Stokes equations. These equations are written in a frame of reference rotating with the blade. This has the advantage of making our simulation not require a moving mesh to account for the rotation of the blade.

The equations that we will use look as follows:

Conservation of mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \vec{v}_r = 0$$

Conservation of Momentum (Navier-Stokes):

$$\nabla \cdot (\rho \vec{v}_r \vec{v}_r) + \rho(2\vec{\omega} \times \vec{v}_r + \vec{\omega} \times \vec{\omega} \times \vec{r}) = -\nabla p + \nabla \cdot \vec{\tau}$$

Where

\vec{v}_r is the relative velocity (the velocity viewed from the moving frame) and $\vec{\omega}$ is the angular velocity.

Note the additional terms for the Coriolis force ($2\vec{\omega} \times \vec{v}_r$) and the centripetal acceleration ($\vec{\omega} \times \vec{\omega} \times \vec{r}$) in the Navier-Stokes equations. In Fluent, we'll turn on the additional terms for a moving frame of reference. For more information about flows in a moving frame of reference, visit ANSYS Help View > Fluent > Theory Guide > 2. Flow in a Moving Frame of Reference and ANSYS Help Viewer > Fluent > User's Guide > 9. Modeling Flows with Moving Reference Frames.

We use the Reynolds Averaged form of continuity and momentum and use the SST k-omega turbulence model to close the equation set.

B. Boundary Conditions

The input values from the experimental data (Emerson & Sinclair, 1967) have been converted to SI units. The values they have used have not been specific, as they tested a number of different shaped propellers and stated only cavitation number but not speed or pressure.

However, they have given a range of 10 to 16 feet per second.

For this project, a fixed linear (upstream) velocity was chosen (13 feet per second) and the revolutions were varied in accordance with J.

$$J = v / (n * d)$$

Where,

J is Advance co-efficient

v is velocity of fluid

n is rotation per sec

d is diameter of marine propeller

The solver required an input of the angular velocity the propeller would be rotating at these values below:

Diameter (m)	Linear velocity (m/s)	J	Revolution (1/s)	Angular velocity (rad/s)
0.4064	3.9624	0.5	19.5	122.52
0.4064	3.9624	0.6	16.25	102.10
0.4064	3.9624	0.7	13.93	87.52
0.4064	3.9624	0.8	12.19	76.58

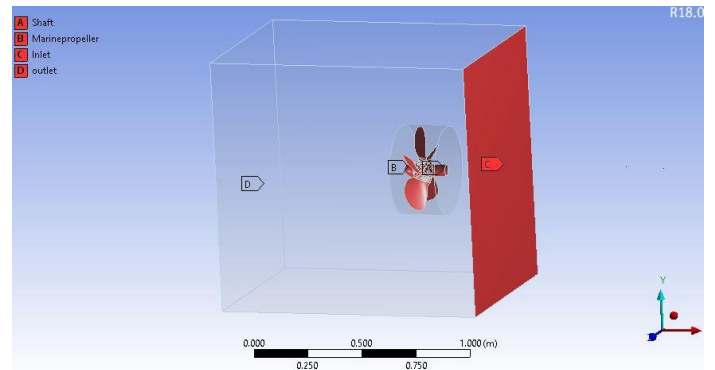


Figure 2 Boundary Condition

VIII. NUMERICAL RESULTS (FOR J = 0.6)

A. Mass Flow Rate

Net results value of mass flow rate at inlet and outlet was checked to see if it makes sense, if mass is balanced

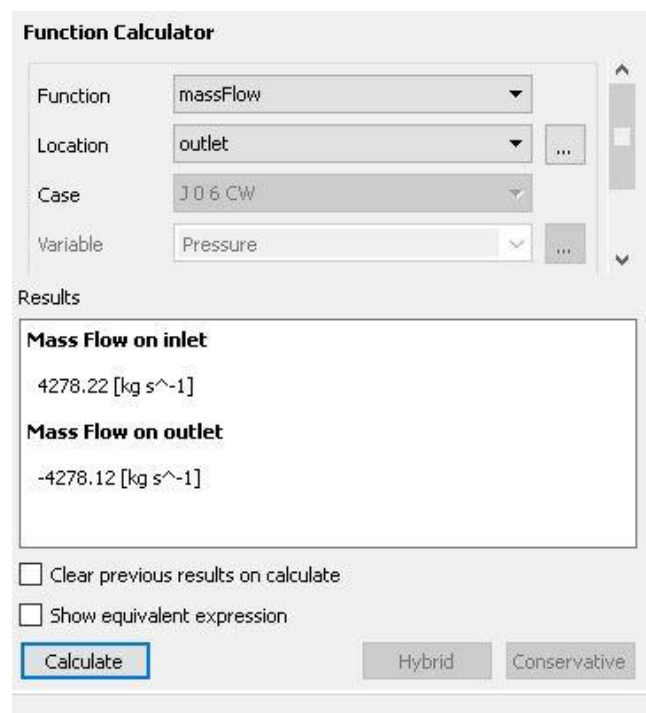


Figure 3 mass flow rate

B. Blade Velocity

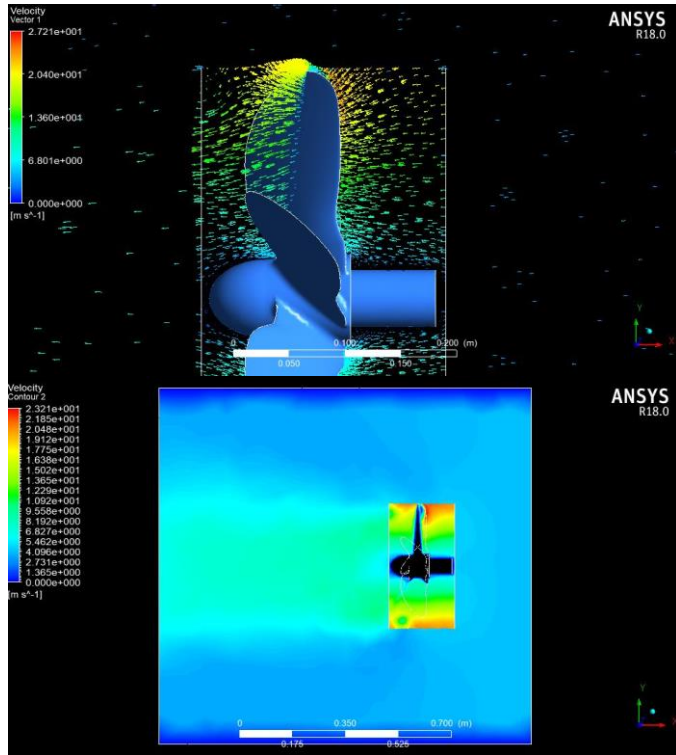


Figure 4 Blade Velocity (Scalar and vector)

C. Pressure Contour

From this plot, it is visible that the pressure is low on back surface of blade as compared to the front surface of blade (realised from colour scheme). Red region shows Positive pressures, green region shows same magnitude but negative in nature, while blue region indicates much higher negative pressure.

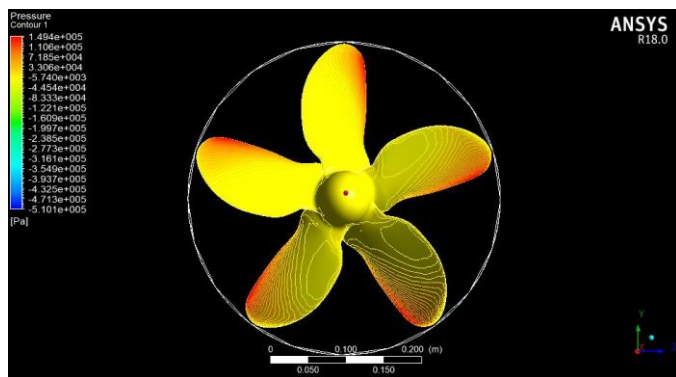


Figure 5 Pressure Contour on Hub side of Blade

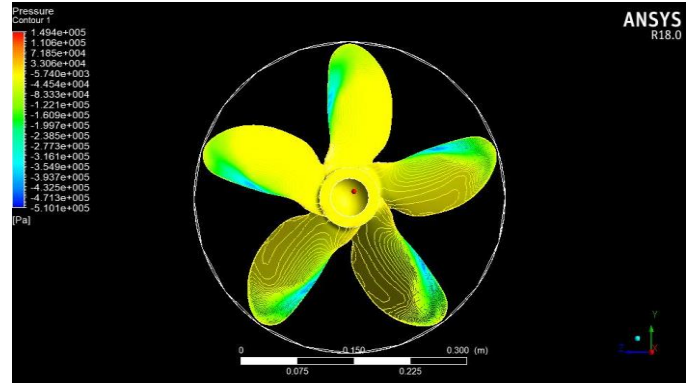


Figure 6 Pressure Contour on Shaft side of Blade

IX. VERIFICATION OF CFD MODEL

A. Comparison with Experimental data

J	CFD Data		Experimental Data K _Q	% Difference
	Torque	K _Q		
0.5	859.138	0.20381	0.22	7.35
0.6	571.102	0.195092	0.189	3.17
0.7	401.792	0.18678	0.15	24.52
0.8	291.9	0.177198	0.12	47.58

Figure 7 Comparison with experimental data

X. FSI SIMULATION

This section considers the deformation due to hydrodynamic loading of a marine propeller blade by performing a steady-state 1-way FSI (Fluid-Structure Interaction) analysis. Previous section of CFD analysis uses ANSYS Fluent to develop the hydrodynamic loading on the blade. In this section, the pressures on the wetted areas of the blade are passed as pressure loads to ANSYS Mechanical to determine stresses and deformations on the blade

A. Setup

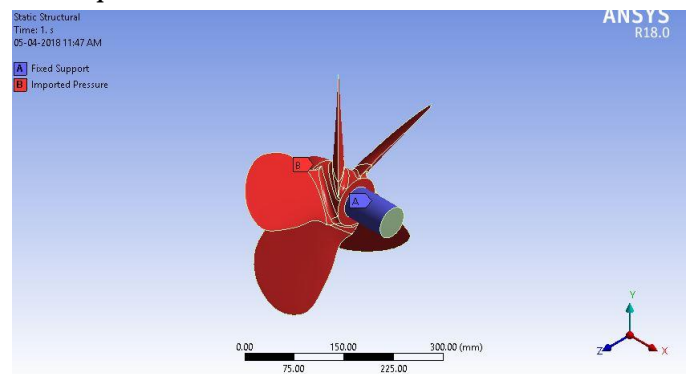


Figure 8 FSI Boundary Conditions

The water flows towards the negative x-direction (into the page on the above diagram) at 3.9624 m/s which is used in experimental procedure. This incoming flow makes the blade rotate at an angular velocity about the x-axis.

We imported pressure over blade surface from CFD model we developed using system coupling. Note that to represent the blade being connected to a shaft.

XI. RESULTS AND CONCLUSION

A. Results

From results, it can be concluded that Polymer fiber based composite material is best suitable replacement from commonly used material viz., Al-alloy. It is also noted that weight is optimized.

Table 1 FSI Results

Sr. no.	Materials	Weight (Kg)	Stress - max (MPa)	Deformation - max (mm)
1	Standard Al-alloy	3.3486	55.347	1.8871
2	Alumina composite	3.3003	55.746	1.9429
3	Carbon composite	2.1156	55.736	3.7884
4	Polymer composite	1.9436	55.7	14.021

B. Conclusion

- Low weight blade can be designed and manufactured with strong load carrying capacity.
- Numerical simulation data of Emerson experimental test.
- Fluid-structure interaction data in said experiment.
- Increase in efficiency of marine propeller on replacement with composite material by reducing its weight without compromising its strength.
- Optimized component helps in increasing performance of marine turbine

ACKNOWLEDGMENT

This work was supported by Department of Mechanical Engineering of Shree L. R. Tiwari College of Engineering, Mira Road, Thane, Maharashtra.

REFERENCES

- [1] Emerson and Sinclair, 1967, "Propeller Cavitation: Systematic Series Tests on 5- and 6-Bladed Model Propellers"
- [2] Colley, Analysis of Flow around a Ship Propeller using OpenFOAM.
- [3] Djahida Boucetta and Omar Imine, 2016, "Numerical Simulation of the Flow around Marine Propeller Series"
- [4] Abolfazl Asnaghi, Andreas Feymark, Rickard E. Bensow, 2012, "Computational Analysis of Cavitating Marine Propeller Performance using OpenFOAM"
- [5] Koyu K., Takafumi K., Akihiko F., Tadashi T., Zhenchuan H., "Study on Unsteady Cavitating Flow Simulation around Marine Propeller using a RANS CFD code"
- [6] Yu-chi Chang, Chin-ning Hu, Jing-chin Tu, Yi-chih Chow, "Experimental investigation and numerical prediction of cavitation incurred on propeller surfaces"
- [7] ZHU Zhi-feng, 2012, "Numerical investigation of cavitation performance of Ship propellers"
- [8] Fang-wen H., Shi-tang D., "Numerical simulation of the structure of propeller's tip vortex and wake"
- [9] Abolfazl A., Andreas F., Rickard B., "Computational Analysis of Cavitating Marine Propeller Performance using OpenFOAM"
- [10] Roberto M., Andrea Di M., Roberto V., "Modeling of vortex dynamics in the wake of a marine propeller"
- [11] M. A. Elghorab, A. Abou El-Azm Aly, A. S. Elwetedy, and M. A. Kotb., "Open Water Performance of a Marine Propeller Model Using CFD"