ISSN 2229-5518

Computational Fluid Analysis Over A Six Blade Hydrodynamic Propeller

CH.RAGHA LEENA[1], D. ANITHA[2], & M. MARY THRAZA[3]
[1,2,3]Assistant Professor, Department of Aeronautical Engineering, Institute of Aeronautical Engineering, Hyderabad, Telangana. India

Abstract— Propeller is used to for thrust generation in this work the hydraulic propeller design and analysis will be carried. It is a known fact that hydro propeller gives the required thrust for a boat to navigate. The present work concentrates on the CFD simulation of a six bladed hydro propeller. A six bladed hydro propeller is designed using CATIA V5 and fluid dynamic simulation is run to capture the flow phenomena during the working. Suitable Shear stress turbulence model will be simulated in ANSYS CFX to capture the turbulence in the flow. Simulations are conducted for different boundary conditions capturing contours of different flow phenomena.

Index Terms— Propeller, thrust, hydraulic propeller, navigation, computational fluid dynamics, turbulence model, simulation.

1 INTRODUCTION

THE propeller can be defined as "a component of a ship-propulsion power plant which converts engine torque into propulsive force or thrust, thus overcoming a ship's resistance to forward motion by creating a sternward accelerated column of water". The blades are normally of NACA airfoil shaped. While in rotation, the blades produce a pressure difference between the forward and rear section of the blade, thus producing a thrust which propels the ship.

The marine propellers are used as standard propulsion mechanism for surface ships and underwater vehicles. Compared to fans and turbines, a marine propeller has a very complex geometry due to its variable section profiles, chord lengths, and pitch angles.

A pressure difference is produced between the forward and rear surfaces of the airfoil-shaped blade, and a fluid is accelerated behind the blade. Propeller dynamics, like those of aircraft wings, can be modeled by Bernoulli's principle and Newton's third law. Most marine propellers are screw propellers with fixed helical blades rotating around a horizontal axis or propeller shaft.

PROPELLER GEOMETRY

Propeller geometry is normally a complicated shape at the edges because of the complex nature of profile. The important terms which determine the shape of the blade profile are:

Propeller Diameter

Diameter of the propeller is two times the distance from the Centre of the hub to the tip of the blade.

Propeller Pitch

Pitch is defined as the theoretical forward movement of a propeller during one revolution -assuming there is no "slippage" between the propeller blade and the water.

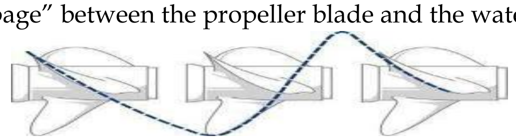


Fig. 1 **Propeller Pitch**

Propeller Rake

Rake is the degree that the blades slant forward or backwards in relation to the hub. Rake can affect the flow of water through the propeller which in turn affects the performance.

Propeller Skew

Skew is the deviation of the blade contour with respect to a radially symmetrical contour.

TYPES OF PROPELLER

Screw Propeller

An actual propeller has blades made up of sections of helicoidal surfaces which can be thought to 'screw' through the fluid.Actually the blades are twisted airfoils or hydrofoils and each section contributes to the total thrust.

Aircraft Propeller

The twisted airfoil shape of modern aircraft propellers was pioneered by the Wright brothers.An aircraft propeller is an aerodynamic device which converts rotational energy into propulsive force creating thrust which is approximately perpendicular to its plane of rotation.

Marine Propellers

Conventional marine propellers remain the standard propulsion mechanism for surface ships and underwater vehicles. Modifications of basic propeller geometries into water jet propulsions and alternate style thruster on underwater vehicles has not significantly changed how we determine and analyze propeller performance.

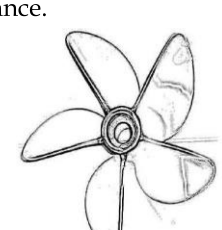


Fig. 2 Marine Propeller

IJSER

ISSN 2229-5518

Variable Pitch Propeller

A variable-pitch propeller or controllable-pitch propeller is a type of propeller with blades that can be rotated around their long axis to change the blade pitch. Constant speed propellers work by varying the pitch of the propeller blades. As the blade angle is increased, it produces more lift (thrust). At the same time, more torque is required to spin the prop, and the engine slows down.

The controllable pitch propellers can be used to run the ship in forward and astern direction both, without the requirement to change the direction of rotation of the engine. The propeller works on the principle of lift generated by each aerofoil section of the blade.

CAVITATION

Cavitation is the formation of vapour bubbles in water near a moving propeller blade in regions of low pressure due to Bernoulli's principle. It can occur if an attempt is made to transmit too much power through the screw, or if the propeller is operating at a very high speed. Cavitation can waste power, create vibration and wear, and cause damage to the propeller. It can occur in many ways on a propeller. The two most common types of propeller cavitation are suction side surface cavitation and tip vortex cavitation.

Suction side surface cavitation forms when the propeller is operating at high rotational speeds or under heavy load. The pressure on the upstream surface of the blade can drop below the vapour pressure of the water, resulting in the formation of a vapour pocket. Under such conditions, the change in pressure between the downstream surface of the blade and the suction side is limited, and eventually reduced as the extent of cavitation is increased. When most of the blade surface is covered by cavitation, the pressure difference between the pressure side and suction side of the blade drops considerably, as does the thrust produced by the propeller. This condition is called "thrust breakdown".

Fig. 3 Cavitation

Operating the propeller under these conditions wastes energy, generates considerable noise, and as the vapour bubbles collapse it rapidly erodes the screw's surface due to localized shock waves against the blade surface.

Cavitation can be used as an advantage in design of very high performance propellers, in the form of the super cavitating propeller. In this case, the blade section is designed such that the pressure side stays wetted while the suction side is completely covered by cavitation vapour. Because the suction side is covered with vapour instead of water it encounters very low viscous friction, making the super cavitating (SC) propeller comparably efficient at high speed. The shaping of SC blade sections however, make it inefficient at low speeds, when the suction side of the blade is wetted.

Cavitation is reduced because the hydrostatic pressure increases the margin to the vapour pressure, and ventilation because it is further from surface waves and other air pockets that might be drawn into the slipstream.

2. METHODOLOGY

Computational fluid dynamics (CFD) is the use of applied mathematics, physics and computational software to visualize how a gas or liquid flows Computational fluid dynamics is based on the Navier-Stokes equations. These equations describe how the velocity, pressure, temperature, and density of a moving fluid are related. Autodesk CFD, formerly Simulation CFD, delivers computational fluid dynamics tools that easily integrate into each phase, and ANSYS workbench. From navier-stokes equation we can derive continuity equation, energy equation, momentum equation.

Thrust:

Thrust is a mechanical force. It is generated through the reaction of accelerating a mass of gas. The gas is accelerated to the rear and the engine is accelerated in the opposite direction. To accelerate the gas, we need some kind of propulsion system.

Equation

If Pe ≠ P0: $F = (me \times Ve) - (m_0^{\cdot} \times V_0) + (Pe - P_0) \times Ae$

If Pe = P0: $F = (me \times Ve) - (m_0 \times V_0)$

But for a propeller, its rotatory motion through the water creates a difference in air pressure between the front and back surfaces of its blades. In order for a propeller blade to spin, it usually needs the help of an engine.

PROPELLER MODELING

CATIA is one of the leading 3D software used by organizations in multiple industries ranging from aerospace, automobile to consumer products. It is a multiplatform 3D software suite developed by Dassault Systems. It is a solid modeling tool that unites the 3D parametric features with 2D tools and also addresses every design-to-manufacturing process.

PROCEDURE:

All the steps are made with definite dimensions which well be useful and easy to learn surface modeling in CATIA V5 and the benefits of surface can also be identified.

IJSER

ISSN 2229-5518

Step 1: Surface tool
In this step we Select Start enter mechanical design select Wireframe and Surface design.
Step 2: Base hub sketch
Diameter of the shaft: 0.160m
Step 3: Extrude
Step 4: Creating a point
Step 5: Creating an Axis
Step 6: Helix.
Step 7: Creating a Reference blade
Diameter of reference blade: 0.720m.
Step 8: Creating a Blade sketch
Here we Create or project the profile on to the extruded surface. Then create two lines closing the loop into a closed sketch.
Step 9: Blade profile thickness
Here we Enter or switch to Part Design. we can generate the propeller blade length of the Shaft: 0.350m

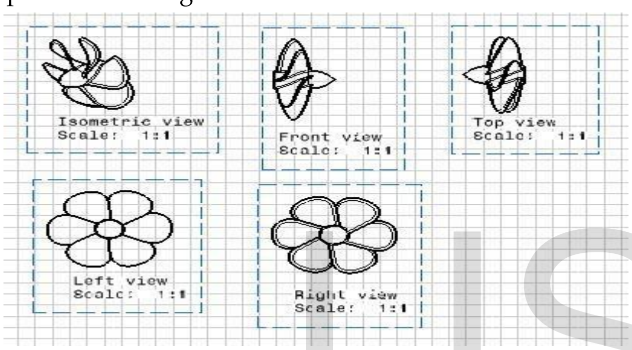


Fig. 4 Drafting of Propeller

IJSER

ANALYSIS OF PROPELLER

A complete computational solution for the flow was obtained using CFX software. Firstly we have to open the Ansys Workbench, by selecting Ansys CFX, then click on geometry and import the .igs format file. Take the measurements in meters and let's assume the enclosure measurements of inlet be ¾th of one meter and the outlet be three meter.

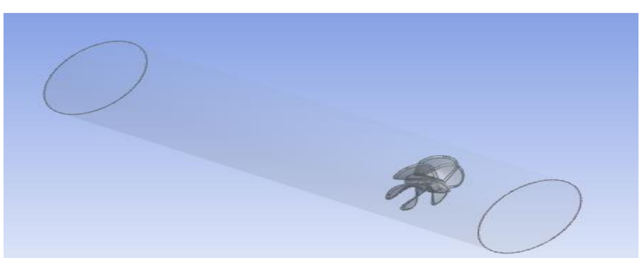


Fig.5 Geometry in ANSYS CFX

MESH

We are changing the mesh default parameters and mesh sizing parameters, such that to get a fine mesh. Give the size function as proximity and curvature, relevance center as medium and curvature normal angle should be 15degrees.

BODY SIZING

Right click on the mesh, then insert the sizing and select the propeller as body then apply on the geometry to give body sizing. Determine the type as sphere of influence, sphere center as global coordinate system, sphere radius as 0.47m and element size as 5.e-003m Click on generate mesh which will take some time to run, then the mesh will be obtained as shown in the figure.

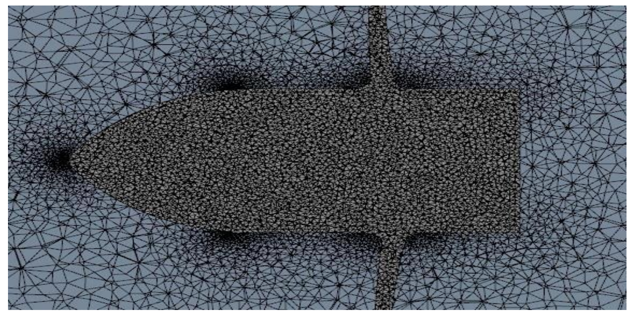


Fig.6 Mesh generated over propeller

BOUNDARY CONDITIONS

Click on setup where the boundary conditions will be represented. Insert domain and select the domain as enclosure which should be of fluid type. Before giving the material library we should define the material properties as sea water and regular water.

Table 1: Fluid Properties

S. No	Fluid Properties	Sea Water	Regular Water
1.	Density (Kg/m3)	1024	997
2.	Molar mass (g/mol)	58.44	18.02
3.	Specific heat capacity (J/Kg K)	3985	4181.7
4.	Temperature (0C)	30	25
5.	Thermal expansivity (K-1)	3.03e-04	2.57e-04

Inlet

The fluid domain would be stationary since the water is flowing through the enclosure while the propeller is rotating. specify the boundary conditions as inlet, outlet and wall by selecting respective surfaces and represent the inlet velocity and outlet pressure and also the turbulence option as K-epsilon (k-ε) turbulence model.

Again insert the domain as solid which is propeller of material library as aluminum which was rotating in X-axis at different rpm and at different inlet velocities will be presented. According to propeller hypothesis, the boat propeller will travel at different velocity incaribean sea water, choppy waters and calm water and their correspondent blade rpm as discussed.

Table 2 Velocity and Blade RPM

S.No	Knots	Velocity (m/sec)	Blade RPM	RPM in rad/sec
1.	25	12.8	1750	183.25
2.	50	25	3500	366.51
3.	80	41	5600	586.43

Outlet

Change the basic settings as outlet section and give the mass and momentum option as static pressure. Now we are taking six cases as three inlet velocities in two different water properties.

3 RESULTS

The final step of the ANSYS CFX is the results obtained of contours, streamlines and calculated pressure and force acting on the propeller.

SEA WATER | **REGULAR WATER**

Inlet pressure = 132020pa
Outletpressure= 180000pa

Inlet pressure = 128540pa
Outlet pressure = 180000pa

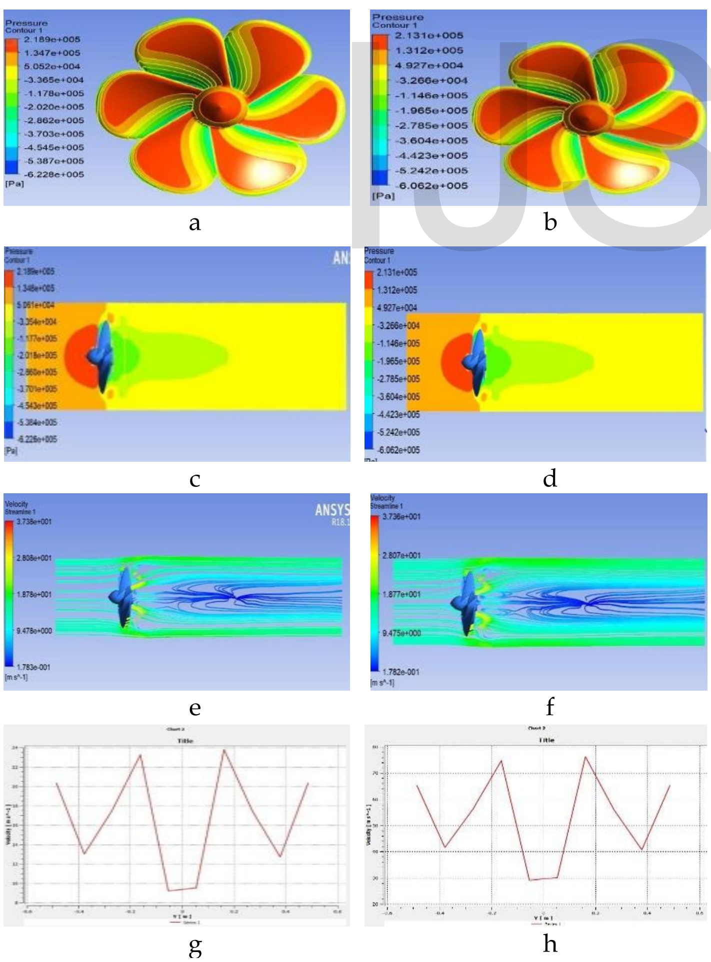


Fig. 7 Pressure contour(a,b), Pressure contour on XY plane(c,d), Surface streamline on XY plane(e,f),and graph(g,h) at velocity 12.8m/s and 1750 RPM

SEA WATER | **REGULAR WATER**

Inlet pressure = 501600pa
Outlet pressure = 700000pa

Inlet pressure= 488400pa
Outlet pressure = 680000pa

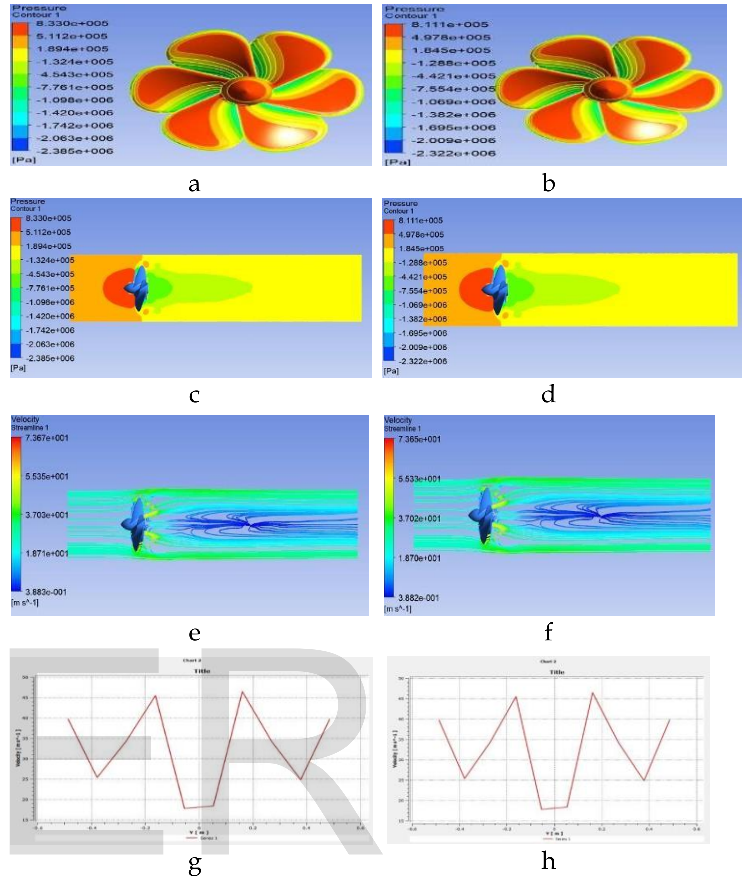
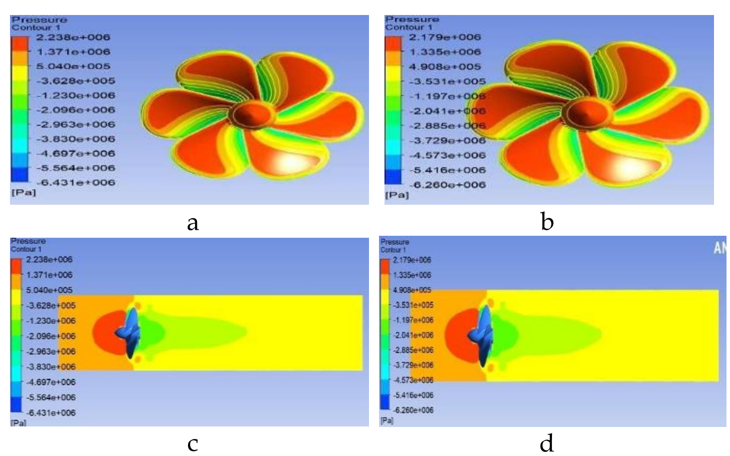


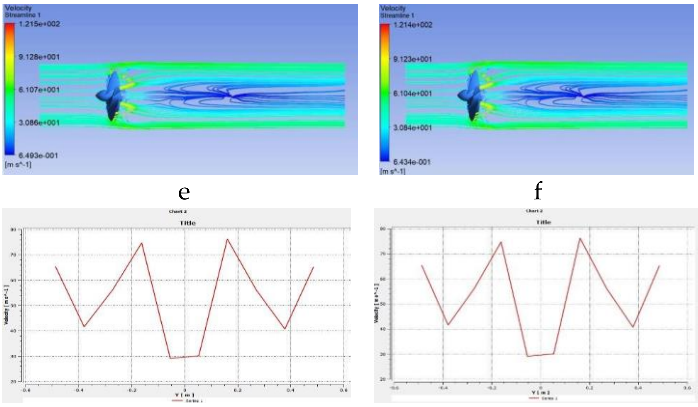
Fig. 8 Pressure contour(a,b), Pressure contour on XY plane(c,d), Surface streamline on XY plane(e,f),and graph(g,h) at velocity 25m/s at 3500 RPM

SEA WATER | **REGULAR WATER**

Inlet pressure= 1346200pa
Outletpressure= 1800000pa

Inlet pressure= 1311600pa
Outletpressure= 1800000pa





e f

g h

Fig. 9 Pressure contour(a,b), Pressure contour on XY plane(c,d), Surface streamline on XY plane(e,f),and graph(g,h) at velocity 41m/s and 5600RPM

Table3

S. No	TYPES	Theoretical values[N]	Obtained values[N]	ERROR (%)
1.	Sea water at 12.8m/s and 1750RPM	146601.45	136267	7.05
2.	Sea water at 25m/s and 3500RPM	570372.71	519093	8.99
3.	Sea water at 41m/s and 5600RPM	1483141.59	1395520	5.91
4.	Regular water at 12.8m/s and 1750RPM	144614.01	132654	8.27
5.	Regular water at 25m/s and 3500RPM	554975.98	505428	8.92
6.	Regular water at 41m/s and 5600RPM	1463725.42	1358540	7.18

IJSER

4 Conclusion

Although In all following six cases, the velocity streamlines from inlet to outlet, the flow has been curled along the wall. On behind flow of streamline the flow has been circulated. When it is observed the temperature variation over the body from front to rare end, accordingly we should select the material of the propeller.On the propeller pressure contour, the half of the blade is high pressure region and remaining half is low pressure region. Due to these, structural damage will occur over the blade but it can resist where it is close to the root. These type of pressure behavior occurred due to one blade behind the other blade.

Pressure contour on the propeller plane says that thrust and mass flow rate varied and the pressure differences are seen in different cases as inlet and outlet pressure. As the same inlet velocity and blade RPM a slight change of pressure difference was observed due to different fluid properties. The theoretical thrust calculated is compared to obtain values shows that the error is varied between 6-9 %. Hence the engine will give the 60% of efficient thrust to the theoretical value.

5 Future Scope

The work is based on pure fluid analysis and next stage we have to adopt the pressure based material or different materials and blade pressure variation will be tested. The theoretical thrust is to be calculated which obtain values shows that the error is to be reduced which increase engine efficient thrust to the theoretical value.

References

[1] Hoshino, T., "Hydrodynamic Analysis of Propeller in Steady Flow Using a Surface Panel Method", Journal of the Society of Naval Architects of Japan, Vol. 165, June 1989, pp. 55-70.

[2] Hanaoka, T., "Fundamental Theory of a Screw Propeller (Especially on Munk's Theorem and Lifting-Line Theory)", Report of Ship Research Institute, Vol. 5, No. 6, 1968, pp. 1-41.

[3] Berchiche, N., Janson, C.-E., 2008, "Grid influence on the propeller open-water Performance and flow field", Schiffstechnik/Ship Technol. Res.55, 87-96, C.-E 2008.

[4] Terje sonntvedt, "Propeller Blade Stresses, Application of Finite Element Methods" computers and structures, vol.4, pp193-204

[5] L. Praveen, S. Anjaneyulu, L.Venugopal "Modelling & Structural Analysis of Propeller Blade", July 2017

[6] N. Vardar, A. Ekerim, "Failure analysis of hydrodynamic propeller blades in a thermal Power plant", 2008

[7] Prof. A.H. Techet Fall., "Hydrodynamics for Ocean Engineers", 2004.

[8] CH. Ragha Leena, Ravi Kumar. P, Anitha.D and P. K Dash, "Experimental Investigation Of Mechanical Characterization Of Pancarbon", International Journal of Applied Engineering Research and Development (IJAERD), Vol. 8, Issue 3, Jun 2018.

[9] M. Marythraza, D. Anitha , P.K. Dash and P.Ravi Kumar, "Vibration Analysis of Honeycomb Sandwich Panel in Spacecraft Structure", International Journal of Mechanical and Production Engineering Research and Development , Vol. 8, Issue 3, June 2018, pp. 849-860.

[10] Ch. Ragha Leena, G. Swathi, and M. Snigdha, "An Experimental Investigation of Annular Fins under Forced Convection", International Journal of Engineering Research and Management, Vol. 03, Issue 12, Dec. 2016.

[11] D. Anitha,G.K Shamili, P. Ravi Kumar, and R.S. Vihar, "Air Foil Shape Optimization using CFD and Parameterization Methods", International Conference of Materials Processing and Characterization, Hyderabad, India, 17 – 19 Mar. 2017.