WWW.IJRPR.COM

International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Rotary Motion Simulation of Marine Propeller Blade using CFD Analysis

[1]Tejas Jadhav, [2]Sameer Bhuvaji, [3]Anmol Khose, [4]Pranjal Wagh, [5]Aniket Kulkarni

[1,2,3,4,5]Student, B.E. Mechanical. D. Y. Patil College of Engineering, Akurdi

A B S T R A C T

At a certain design speed, a marine propeller produces enough thrust to move a ship. In order for a ship to operate at its best, its propellers are essential. With an appropriately designed propeller, it is taken into account to match the engine's power and shaft speed with the ship's size and operating speed. The flow around a marine propeller is challenging due to the complicated geometries of the propeller. In general, expensive experiments like open-water and self-propulsion model tests that are conducted early in the design process are used to determine and analyze the performance characteristics of a propeller. In this study, computational fluid dynamic analysis was used to investigate the hydrodynamic performance and parameters of marine propellers. In this paper, a propeller model is developed with respect to some design constraints such as ship speed, vessel draft, etc., and analyzes the performance using CFD tools in ANSYS 2023 R1 Student. In the case of simulation, the existing material such as Aluminium, Bronze, and Nickel-Aluminium Bronze alloy is used for rotary simulation. Finally, we got different results for different materials.

Keywords:- Ship, Marine Propeller, CFD, Material, ANSYS 2023 R1.

1. INTRODUCTION

A ship's or boat's thrust-generating system is known as its marine propulsion system. Propulsive efficiency is one of the most important factors to take into account when thinking about ship designs and design improvements. For the ship to operate efficiently, propellers are required. In the pre-design stage, full-scale observations and model testing are required to predict and determine the propulsive efficiency of ship designs. Additionally, a variety of strategies and analyses have been developed by researchers and naval architects to achieve increasing propulsive efficiency through comparison and investigation of ship hulls, propellers, rudders, and energy-saving technologies. The need to increase ship efficiency is being driven by high marine fuel costs and low freight rates. Advanced ship propulsion options are one way to make significant advancements in this area, but they call for extremely meticulous concept, design, and construction procedures. The only reliable method of identifying and assessing the characteristics of the hull-propeller interaction, the powering performance, and the propulsion parameters of the ship were traditionally through model tests of self-propulsion. Numerical simulations of ship self-propulsion have recently drawn more attention due to the quick developments in the field of computational fluid dynamics (CFD). Making computational fluid dynamics (CFD) analyses of self-propulsion calculations for ships with propellers is now possible thanks to the increased capabilities of numerical flow simulations.

Nomenclature

CFD: Computational Fluid Dynamics

1.1 Model of Propeller

The creation of a 3D propeller model is the initial stage in the CFD analysis. A 3D propeller model is developed for analysis as shown in fig. 1 and the geometric properties of the model propeller are shown in Table.1 and Table. 2.

Fig. 1 CAD Model of Propeller

Development Process

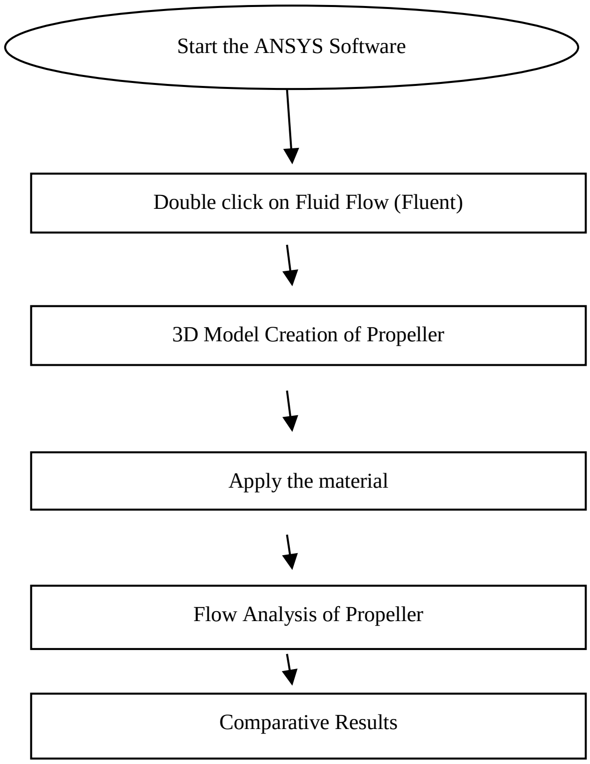

Table 1: Particulars of a 3-Bladed propeller.

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

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	0.052	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 the leading edge and generator line at r.

B= Distance between the leading edge and location of maximum thickness.

C= Chord length of blade section at radius r.

T= Maximum blade section thickness at radius r.

1.2 Structure

The most advanced computer programmer for simulating fluid flow, heat transfer, and chemical reactions in complex geometries is called ANSYS Fluent. ANSYS Fluent offers comprehensive mesh flexibility, enabling you to address your flow issues with unstructured meshes that can be produced relatively easily around complex geometries. Mesh types that are supported include 2D triangle and quadrilateral, 3D tetrahedral, hexahedral, pyramid, wedge, and polyhedral meshes, as well as mixed (hybrid) meshes. We can additionally fine-tune or coarsen our mesh based on the flow solution using ANSYS Fluent. The ability to produce mesh using Fluent's meshing mode for 3D geometries is also described. The created mesh may be read into ANSYS Fluent. Setting boundary conditions, specifying fluid characteristics, running the solution, fine-tuning the mesh, post-processing, and visualizing the results are all carried out within Fluent's solution mode.

The fundamental procedural stages are listed below and are followed after determining the key elements of the problem that you're trying to solve:

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.

1.3 Defining Model Geometry and Meshing

The method begins with importing the step-formatted propeller geometry into Ansys Design Modeller, where the domain is produced as seen in Fig. 2. It uses the x, y, and z coordinates of the Cartesian coordinate system. Positive directions are upstream, to the port, and downstream, and the origin was in the hub's center. In Fig. 3, the domain dimensions were displayed. There were two sections in the solution field: global and sub-domain. The Coriolis acceleration terms are used in the governing equations for the fluid in the subdomain frame to approximate the spinning of the propeller. The sub-domain frame is enclosed by the global frame.

An essential stage in the simulation process is creating a proper and fine grid. The key concerns for creating the mesh are promoting minimal inaccuracy in the simulation process and cutting down on the program computing time. There are two primary ways to create the mesh: one uses an algebraic technique to create the grids, and the other uses partial differential equations to create the grids. It is feasible to achieve a high degree of automation in generation with unstructured grids since there are no restrictions on where points may be placed. These propeller models use an unstructured mesh, also known as an auto volume mesh, typically created after configuring all the parameters, including inflations, growth rates, face and contact sizes, and body and body sizes. Although this type of mesh will lower human costs, it will put a lot of strain on the computer's CPU.

So the main problem with mesh creation here is to reduce the very long computing time for each simulation. Since the models for the propellers are three-dimensional, the analysis of the three-dimensional models uses a three-dimensional mesh.

Table 3: Mesh Options and their Position

Mesh	**Position**
Mapped Face Meshing	Outer Wall of Main Domain
Patch Conforming Method	Main Domain

Face Sizing	Propeller Blade
Inflation	Outer Wall of Main Domain

1.4 Solver Setup

Prior to executing the solutions, the setup must be reviewed and adjusted once all modifications and tweaks to the meshing have been made. This is done in the following ways:

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

1.5 Results

The continuous information in the precise solution of the differential equation defining the physical model would have been substituted with discrete values, resulting in all grid nodes having interaction qualities with their neighbors. In the FLUENT program, a finite volume approach was utilized to transform the governing equations into algebraic equations that could be solved numerically.

CFD Analysis Enclosure:

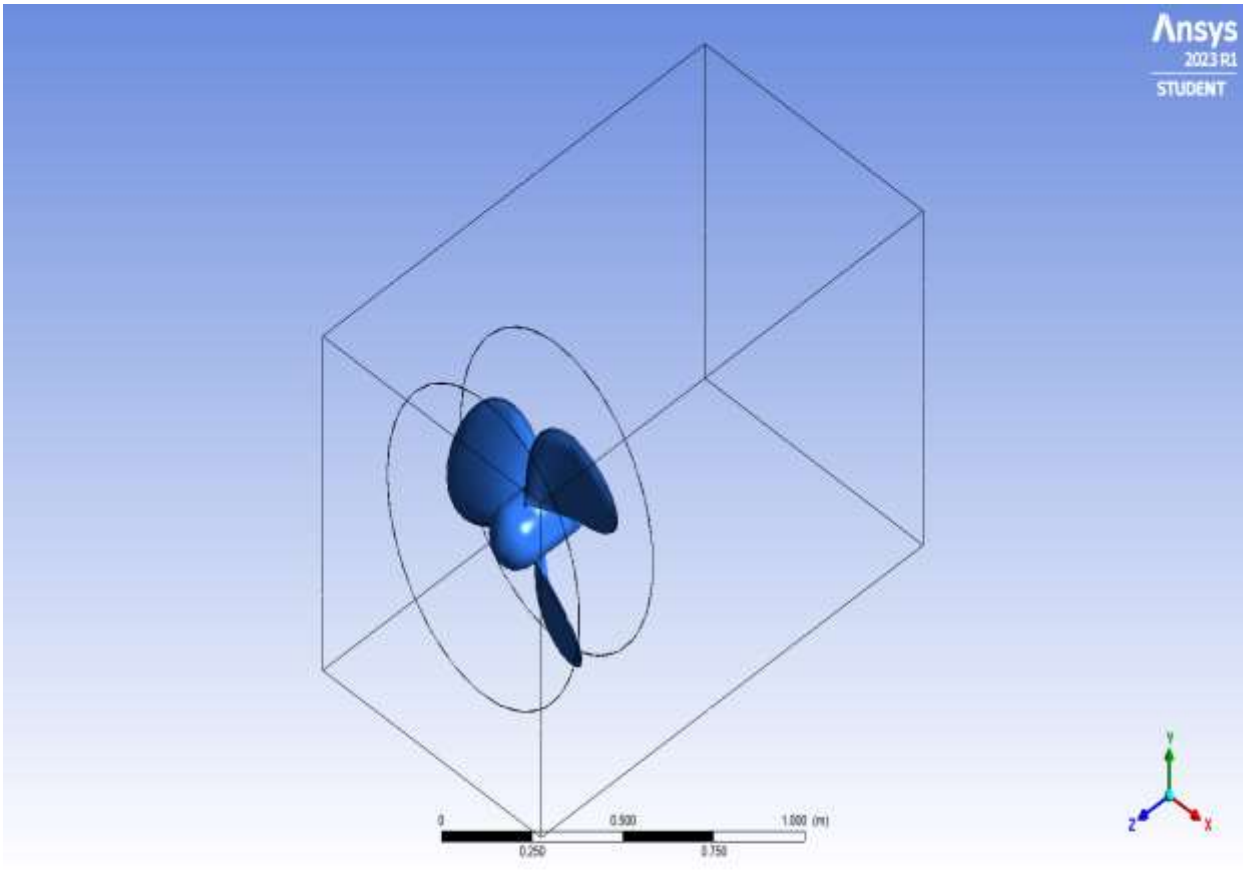

Fig. 2 Enclosure for CFD Analysis

Pressure Gradient:

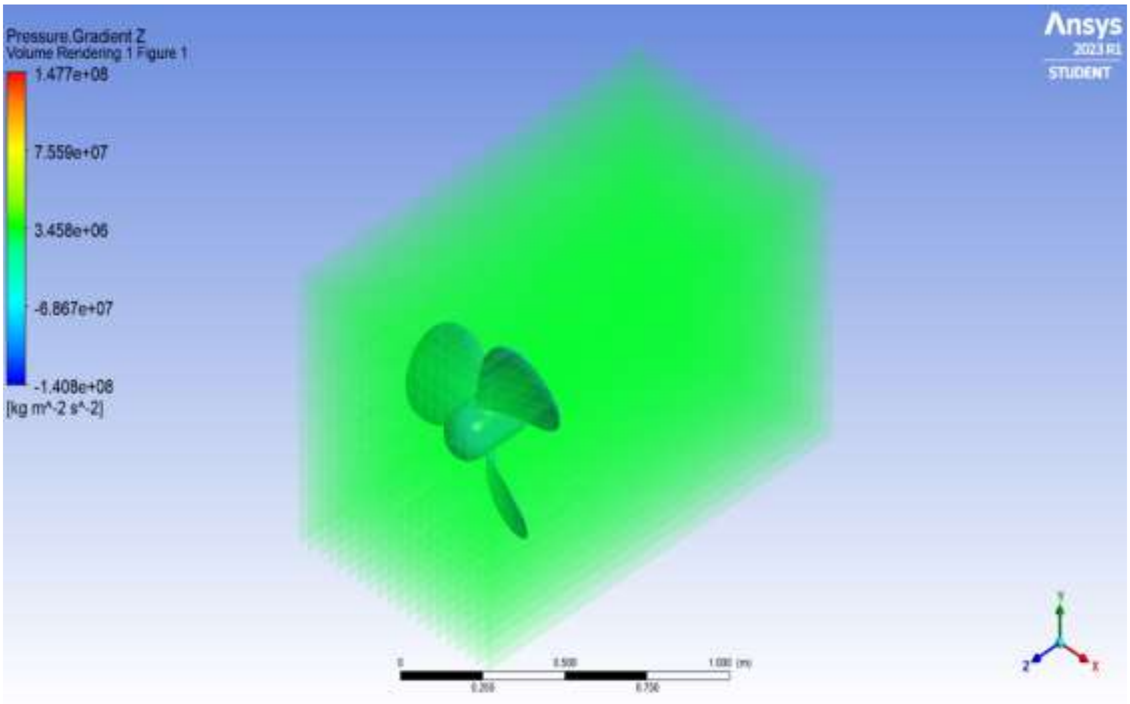

Fig. 3 Aluminum Pressure Gradient

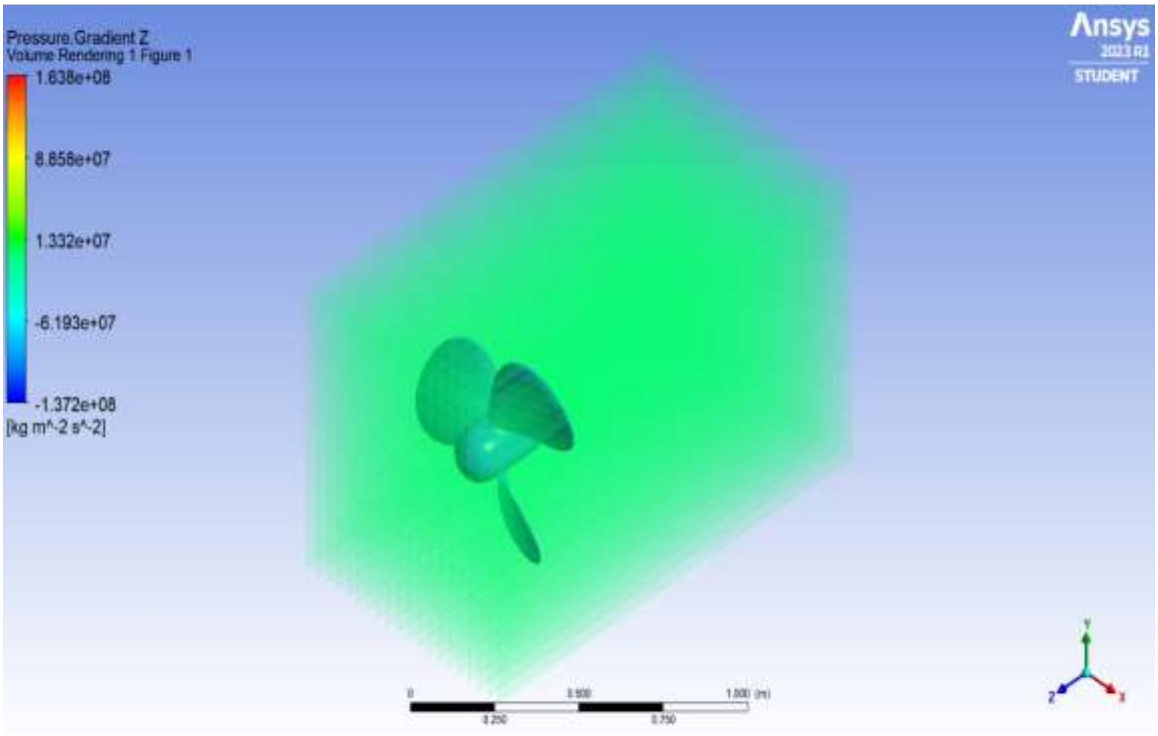

Fig. 4 Bronze Pressure Gradient

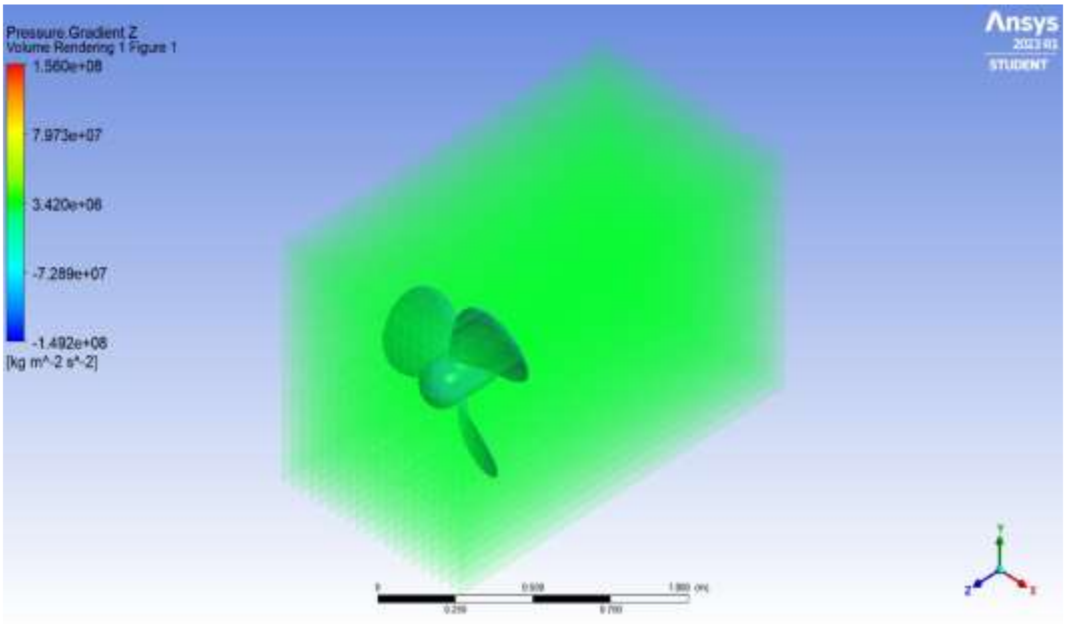

Fig. 5 Nickel-Aluminium-Bronze Pressure Gradient

Velocity Gradient:

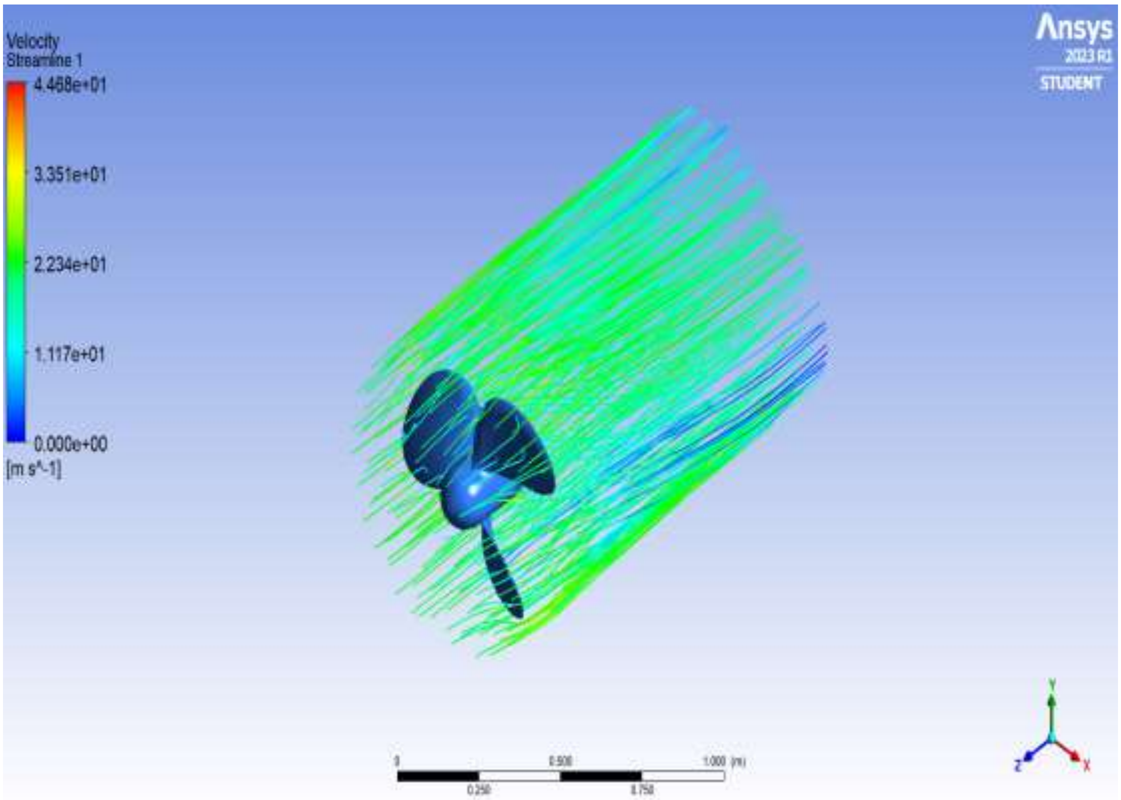

Fig. 6 Aluminium Velocity Gradient

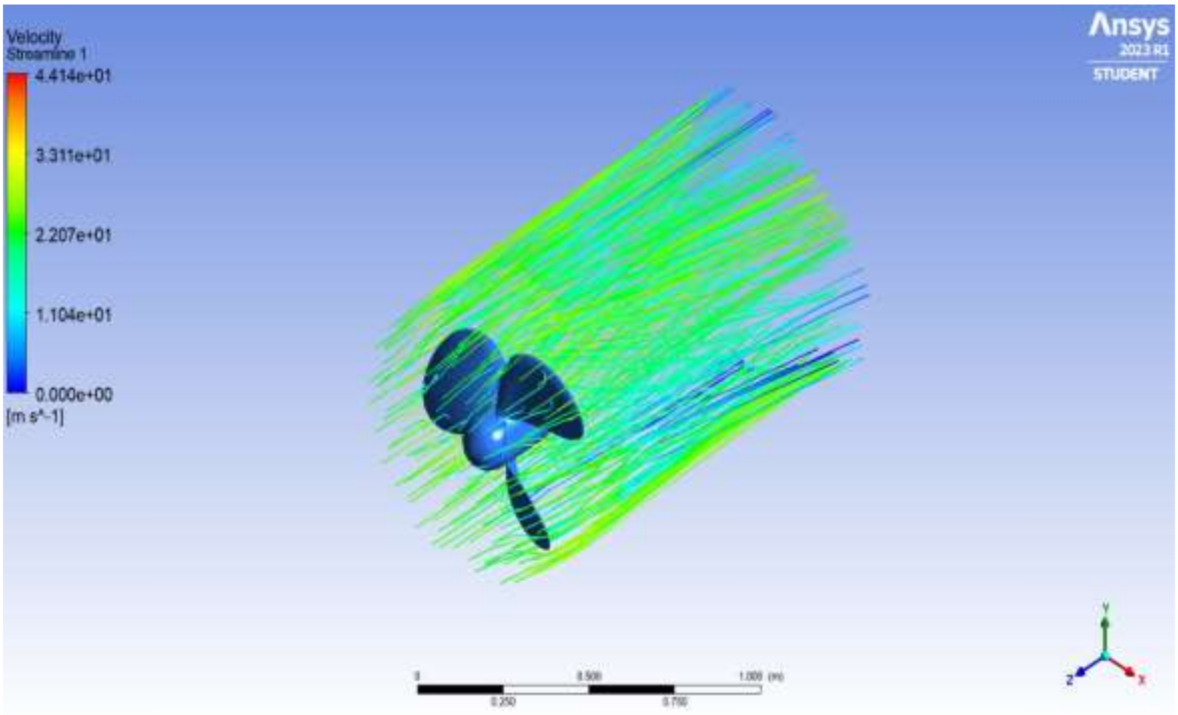

Fig. 7 Bronze Velocity Gradient

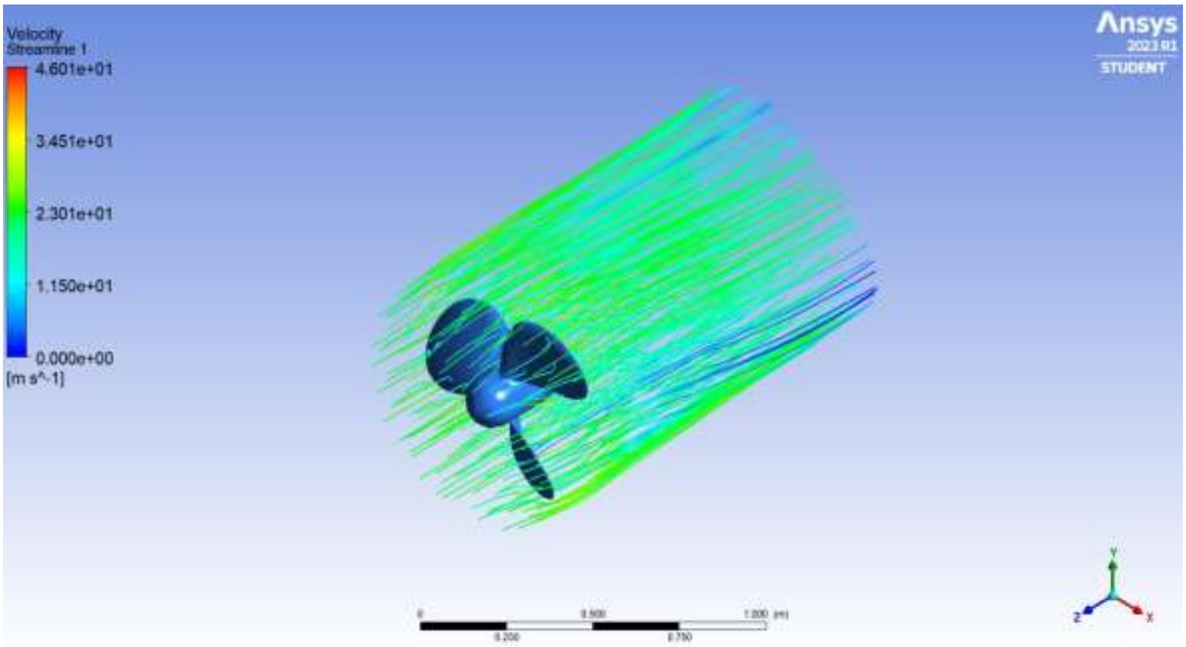

Fig. 8 Nickel-Aluminium-Bronze Velocity Gradient.

1.6 Conclusion

In conclusion, our paper focused on the simulation of marine propeller blade rotation using CFD analysis. We used a three-blade propeller model built on Fusion 360 and imported its geometry into ANSYS for computational fluid dynamics simulations. By analyzing three different materials: aluminum, bronze, and nickel-aluminum-bronze alloy, we investigated the effects of flow velocity and pressure gradient Through our study, we found that the pressure gradient surface was highest for aluminum materials, while nickel-aluminum-bronze composite was the lowest of the three. The flow velocities of the materials showed slight variation, ranging from 4.4 to 4.6 m/s. These findings indicate that the choice of materials significantly affects the pressure gradient but little affects the flow velocity. Such studies can contribute to the design and optimization of marine propeller blades considering the desired trade-off between pressure gradient and flow velocity for a specific application Future research could investigate other factors and develop the simulation parameters well again to make the propeller blade performance analysis more accurate and comprehensive.

References

1. Vijayanandh R, Venkatesan K, Ramesh M, Raj Kumar G, Senthil Kumar M, Optimization of Orientation Of Carbon Fiber Reinforced Polymer Based On Structural Analysis, International Journal of Scientific & Technology Research, ISSN 2277-8616, Volume 8 - Issue 11, November 2019.

2. Vijayanandh R et al., Fatigue life estimation of Aircraft engine compressor with suitable material selection" At International Conference on Intelligent Systems and Control (ISCO2016), IEEE digital explore, ISBN number of 978-1-4673-7807-9, 978-1-4673-7805-5, and 978-1-4673-7806-2 (November 2016), DOI: 10.1109/ISCO.2016.7727055.

3. Vijayanandh. R, Senthil Kumar. M, Sanjeev Kumar. B, Akshaya. V, Nishanth. B & Sindhuja. K, Numerical Study on Drag Effect of Waste Collector Attachment in the Train, International Journal of Mechanical and Production Engineering Research and Development, ISSN(P): 2249-6890, Vol. 8, Special Issue 7, Oct 2018, pp. 1060-1078

4. Senthil Kumar M et al., Conceptual Design and Comparative Computational Analysis of Secondary inlet of Rotary-wing Aircraft Engine, Journal of Advanced Research in Dynamical and Control Systems, Vol. 9. Sp– 14 / 2017, pp 1189 – 1209.

5. M. Senthil Kumar, R. Vijayanandh, N. Kaviarasan, R. Dinesh Kumar, I. Adrin Issai Arasu and R. Kanmani raja, Computational and Theoretical Analysis of Aerodynamic Performance on Roller Airfoil, October 2018, International Journal of Engineering & Technology, 7 (4.10) (2018), pp. 637-642, DOI:10.14419/ijet.v7i4.10.21302.

6. Raj Kumar G, Balasubramaniyam S, Senthil Kumar M, Vijayanandh R, Raj Kumar R, Varun S, Crash Analysis on the Automotive Vehicle Bumper, International Journal of Engineering and Advanced Technology, ISSN: 2249 – 8958, Volume-8, Issue-6S3, 2019, pp. 1602 - 1607, DOI: 10.35940/ijeat.F1296.0986S319.

7. Vijayanandh R, Senthil Kumar M, Rahul S, Thamizhanbu E and Durai Isaac Jafferson M, Conceptual Design and Comparative CFD Analyses on Unmanned Amphibious Vehicle for Crack Detection, Lecture Notes in Civil Engineering, ISSN: 2366-2557, book title " Proceedings of UASG 2019", Book Subtitle : Unmanned Aerial System in Geomatics, eBook ISBN: 978-3-030-37393-1, DOI:10.1007/978-3-030-37393-14.