Design of Experiment and Computational Fluid Dynamics Used to Optimize Hydrodynamic Characteristics of the Marine Propeller

Rohit Suryawanshi

***Abstract*—**In this study, the commercial Computational Fluid Dynamics (CFD), ANSYS-Fluent, has been used to optimize the marine propeller with the design of experiment (DOE) method. At the initial stage, different propeller parameters ware selected for the three different levels. The four characteristics factors are: no. of the blade, camber value, pitch delta & chord at the hub. Then, CAD modelling is performed by considering the selected factor and level. In this investigation, a total of 9 test models are simulated with the Reynolds-Averaged Navier-Stokes (RANS) equations. The standard, realizable k-ω Turbulence Model & Multiple Reference Frame (MRF) were used to observe the propeller's thrust. The relationships between thrust and propeller characteristics were investigated to discover optimum propeller for a ship from the obtained results.

***Keywords*—**Marine propeller, Computational Fluid Dynamics, optimization, DOE, propeller thrust.

I. Introduction

THE predicting the propeller's hydrodynamic characteristic through the propeller's fluid becomes the most challenging thing in the CFD sector. The complex geometry of marine propellers, mesh grid generation, and turbulence modelling remain the main problems instead of the numerical simulations [1]. There are lots of theories explaining about working of thrust produces by the propeller. However, all theories' working principle is simple, but mathematics is quite complex, so several assumptions were made. Banik et al. [2] investigate the propeller's hydrodynamic characteristic with, induction factor method based on normal induced velocity. From the investigation, it was clear that the average induced velocity can be obtained by accurately employing the induction factor. These procedures are used to determine the propeller characteristic to satisfy with the experimental data. Samir et al. [3] carried out the geometric configuration to optimize the hydrodynamic propeller. They perform a numerical simulation of $k-\varepsilon$ and the $k-\omega$ SST on the various propeller model. The final result concluded that each model gives an acceptable level of accurate results of the propeller. Loi et al. [4] presented the effects of the rudder on hydrodynamic performances of the propeller in both cases of the propeller with and without a rudder. Additionally, the relationship between blade pitch angle and hydrodynamic performance is also examined for propeller optimization. The final results conclude that the hydrodynamic characteristic of the propeller slightly changes with users of the rudder. Also, blade pitch plays an essential role as blade pitch goes up thrust and torque coefficient increases drastically. Saha et al. [5] study the development of the B-series propeller model with CFD tools. The simulation effect is considered to investigate the relationship between the thrust effect and the advance coefficient. Finally, the numerical result compared with the experimental data shows that CFD values are always slightly higher than the empirical one. Hollenbach et al. [6] explore all possibilities to optimize the propeller using recent trends. The twin-screw appendages, propulsion improving devices (PID), the pre-swirl stator of Daewoo Shipbuilding & Marine Engineering Co. Ltd (DSME), the Thrust Fin of Hyundai Heavy Industries Co. Ltd (HHI), Post Stator of Samsung Heavy Industries Co. Ltd (SHI), safer fins of SHI, and lots more. Author explained working of propeller numerically and also with the experimental report.

Rohit Suryawanshi is with the University of Mumbai, Mumbai, 400032, India (phone: 9619023741; e-mail: rohitsuryawanshi1520@gmail.com).

Recently, global warming is a significant concern, and it is necessary to reduce the CO_2 level to control global warming; so, to address this issue 'Paris Agreement for shipping' is held in April 2018. The International Maritime Organization (IMO) came with the ambition to reduce CO_2 emissions by at least 50% by 2050. Several studies were conducted on a marine propeller to minimize CO_2 emission such as pre-swirl stators, skew, and rake angles: the rake and skew angle of the concept are illustrated in Fig. 1. Zondervan et al. [7] investigate the pre-swirl stator effect on the single and twin-screw ship. The focus of the studies was on reducing fuel consumption. The rotational losses induced by the pre-swirl stator were investigated along with viscous effect.

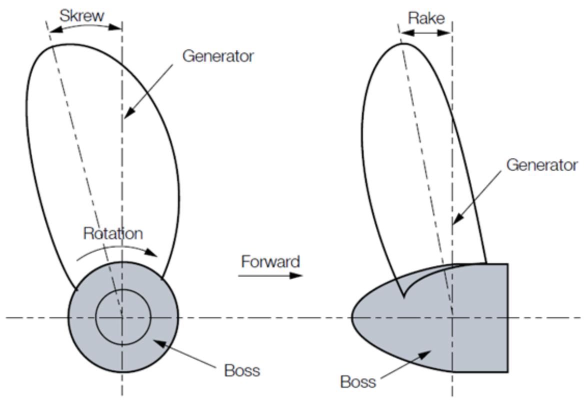


Fig. 1 Rake and skew angle

II. METHODOLOGY

The methodology used in this paper is as follow:

- The four different factors and three levels were selected for the DOE optimization.
- The CAD modelling was done with the help of CAESES and SolidWorks software.
- CFD was performed on all nine tests, and results were extracted from the Ansys software's post-processing.
- The optimal combination was selected based on the final results.

III. PROPELLER MODELLING

A. Basic Approach

For the CFD simulation, the 3D model was necessary, so the first step is to design the propeller's 3D model. The CAESES software is initially used to model the propeller with all other characters like a number of blades, camber value, pitch delta & chord at the hub to generate the propeller's geometry. Later, SolidWorks was used to generate the MRF region and flow region for further flow simulation. The general 3D model of the propeller is shown in Fig. 2.

Fig. 2 3D model of propeller

Detailed geometric properties of a particular propeller are given in Table I and different characteristics of the marine propeller are illustrated in Fig. 3.

TABLE I
GEOMETRIC PROPERTIES OF PARTICULAR PROPELLER

Parameter	Dimension
Number of Blades	3
Propeller Diameter	4 m
Skew Tip	0.2
Camber Value	0.03
Chord at Hub	0.30
Pitch Delta	0.1

B. Defining Model Geometry & Meshing

At the beginning of the process, the propeller geometry is imported into ANSYS. The Cartesian coordinates system is used. The Boolean operations are used for combinations of geometric entities and also used to generate two separate regions. The mesh generation is one of the crucial steps in the simulation process. The finer mesh is used for simulation to capture all possible flow around the propeller and within the MRF region [8].

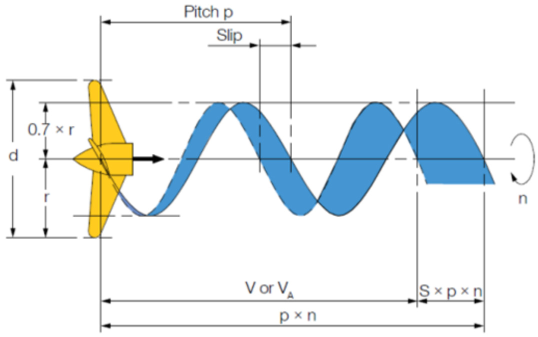


Fig. 3 Different characteristic of the propeller

C. Simulation Setup

The pressure-based solver is used in this study along with absolute velocity function by considering transient time condition. For model properties, we set the viscous model as k-epsilon (2 equations), realizable k-epsilon model, and scalable wall function treatment near the wall [9] as the propeller is submerging in sea-water, the flow medium is chosen as water with 1000 kg/m^3 of density. The moving reference frame (MRF) is used to a rotating frame of reference that modifies the rotating zone's governing equations. The MRF region rotated at 600 RPM. The details of the cell zone are shown in Table II.

TABLE II
CELL ZONE CONDITIONS

Domain	Motion	Frame Motion
Sub-Domain	Relative to Cell Zone	Absolute
	Rotation-Axis Origin	0,0,0
	Rotation-Axis Direction	0,0,1
	Speed (RPM)	600
Main Domain	Motion	Stationary

The 10 m/s is selected for the inlet boundary condition with a reference frame is absolute. The details of boundary condition of the simulation set-up are shown in Table III.

TABLE III
BOUNDARY CONDITIONS

Boundary Conditions type	Condition	Value
Velocity Inlet	Reference Frame	Absolute
	Velocity Magnitude	10 m/s
Pressure Outlet	Backflow Reference Frame	Absolute
	Gauge Pressure	0 Pascal
	Backflow Direction	Normal to Boundary
Outer wall	Wall Motion	Stationary
	Shear Condition	No Slip
Propeller Blade	Wall Motion	Stationary
	Shear Condition	Slip

D. Selected Factor & Level for DOE

Before the experimental test, defining the right approach in the design stages was necessary. Determining the number of levels, different factors, and nature of the expected result were required. The paper's sole approach was to optimize the propeller, which can be accomplished by maximizing the thrust generated by the propeller. Thus "Larger the better" approach was adopted for the DOE [10]. The factor & level considered for the testing were given in Table IV.

TABLE IV
FACTOR AND LEVEL SELECTED FOR ORTHOGONAL ARRAY

Factor	Level 01	Level 02	Level 03
Number of Blades	3	4	5
Camber Value	0.03	0.04	0.05
Pitch Delta	0.1	0.2	0.3
Chord at Hub	0.3	0.35	0.4

IV. Results and Discussion

In this section, the CFD results of hydrodynamic performances of the propeller were explained with TEST 2 model and shown in Fig. 4. The principle of pressure distribution on the two faces of the blade satisfies the axial turbomachinery's theoretical law. There is a pressure difference between the pressure face and the propeller's back face in operation, and that difference makes the propeller thrust overcome the ship hull resistance. The pressure distribution on the two faces of the blade mainly depends on the velocity inlet and 10 m/s is considered for inlet velocity boundary condition, and almost all the blade area has the pressure value of about 4.7×103 Pa. In contrast, almost all suction face areas have pressure in the range of -1.46×104 Pa.

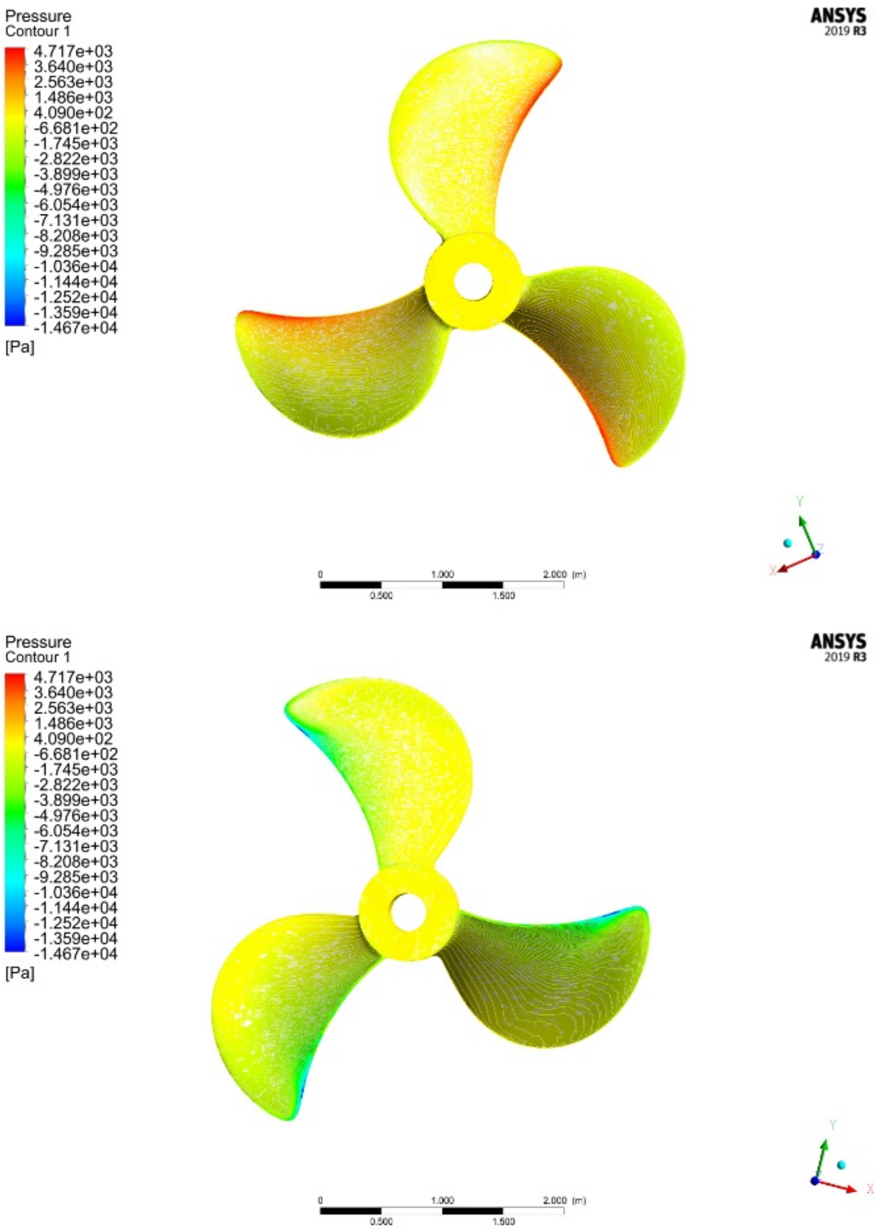
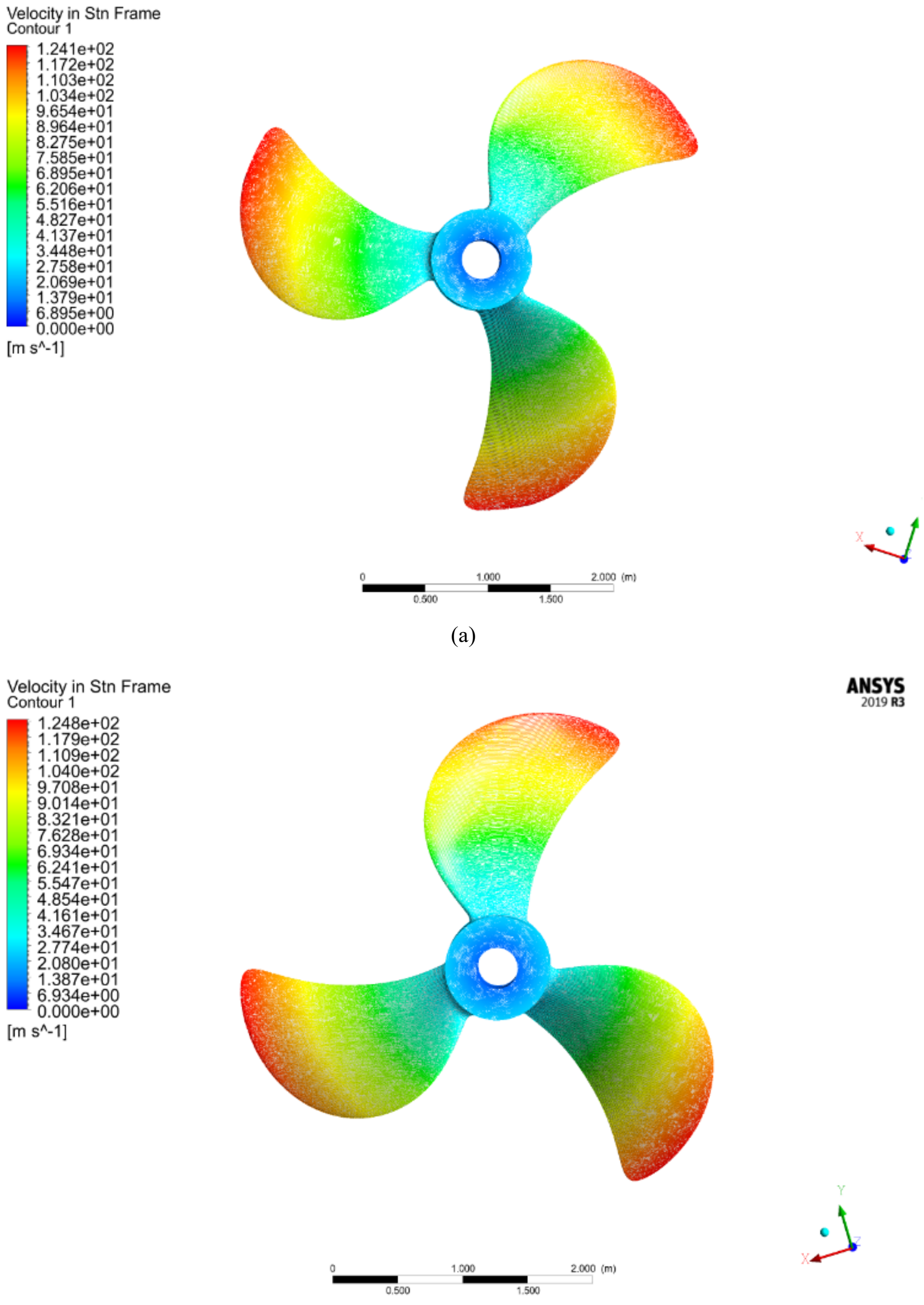


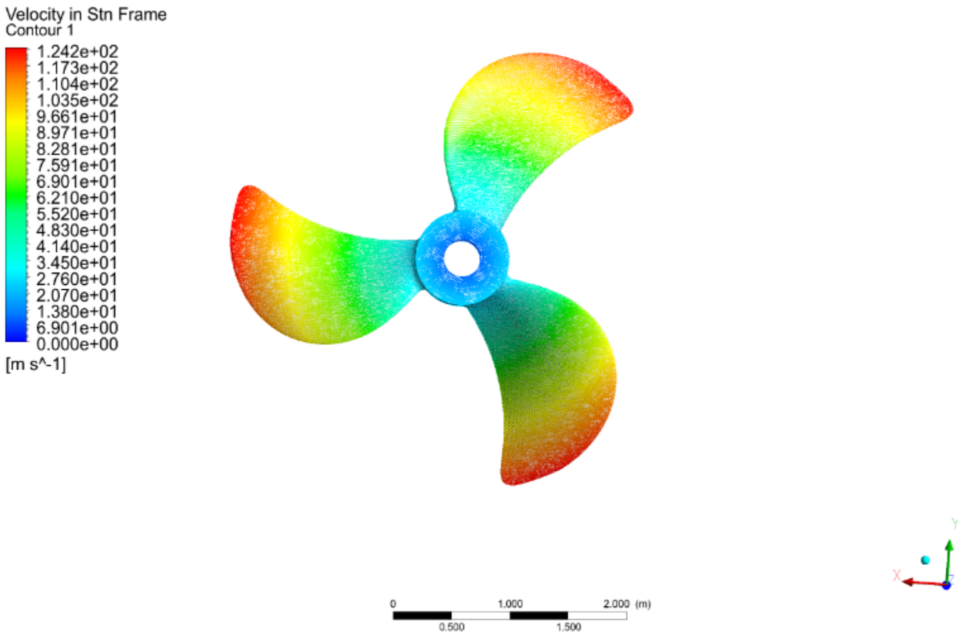
Fig. 4 Pressure distribution on propeller

The CFD outcomes show that the fluid accelerates as it approached the propeller due to low pressure in the propeller front. Nevertheless, the water continues to accelerate when it leaves the propeller. The velocity distribution on all nine tests of marine propeller surface is shown in Fig. 5.

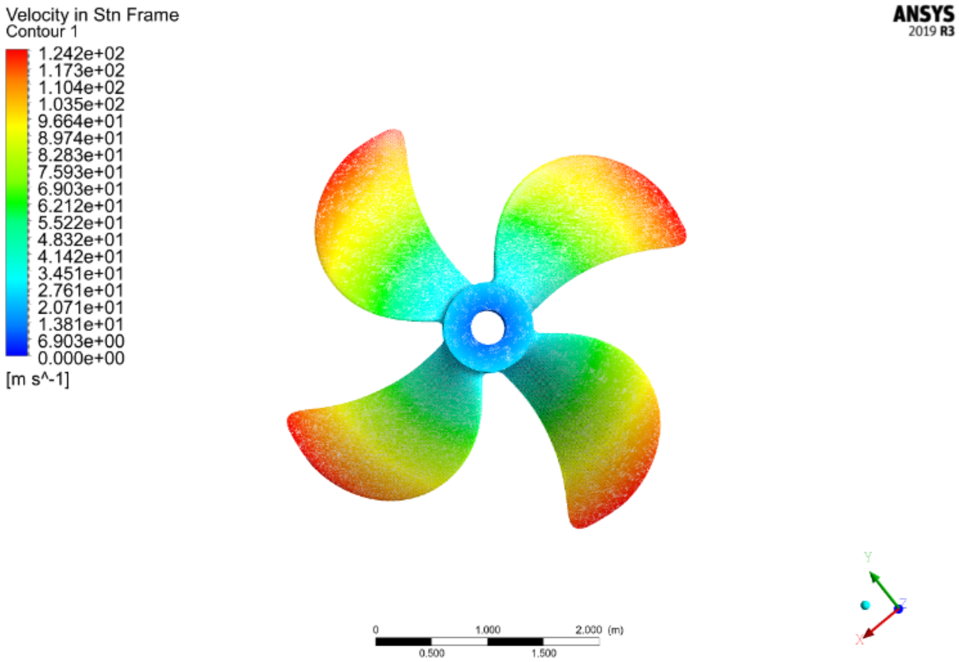


(a)

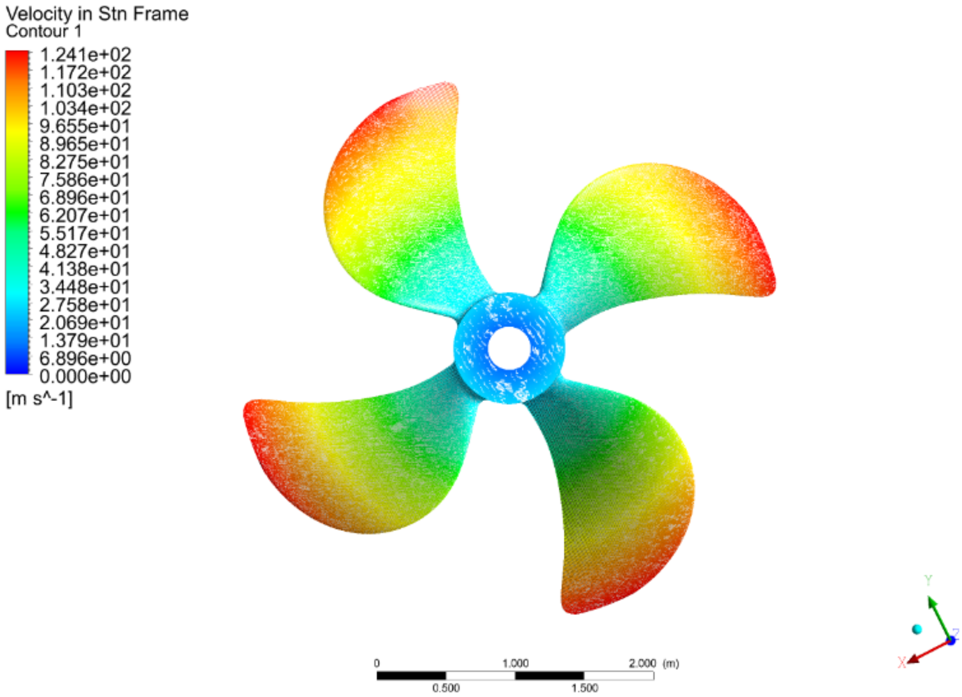
(b)


Velocity in Stn Frame
Contour 1
1.242e+02
1.173e+02
1.104e+02
1.035e+02
9.661e+01
8.971e+01
8.281e+01
7.591e+01
6.901e+01
6.210e+01
5.520e+01
4.830e+01
4.140e+01
3.450e+01
2.760e+01
2.070e+01
1.380e+01
6.901e+00
0.000e+00
[m s^-1]
0
1.000
2.000 (m)
0.500
1.500

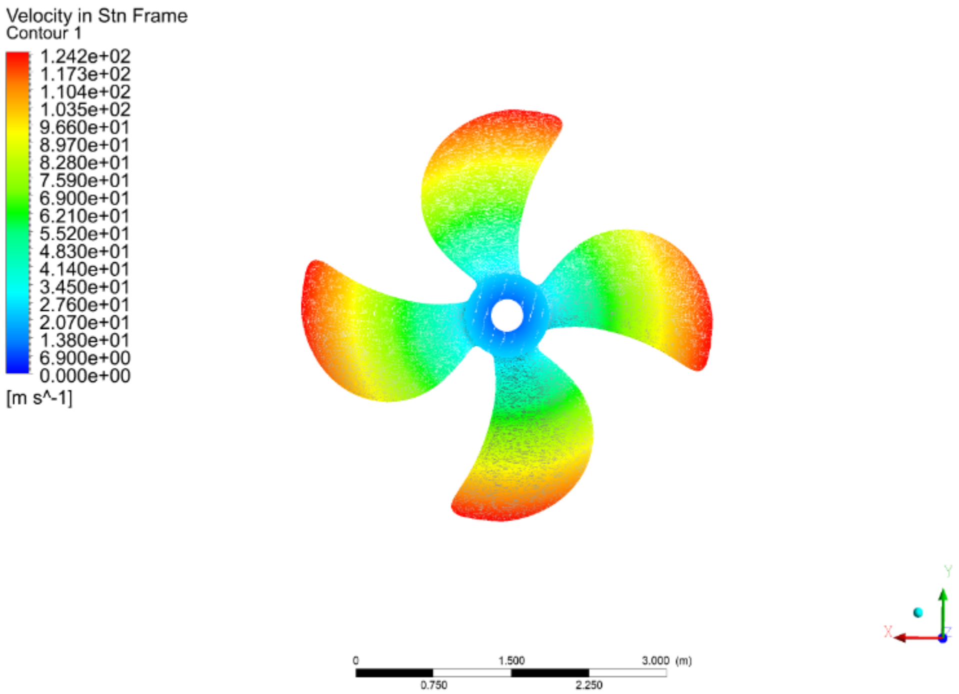
(c)


Velocity in Stn Frame
Contour 1
ANSYS
2019 R3
1.242e+02
1.173e+02
1.104e+02
1.035e+02
9.664e+01
8.974e+01
8.283e+01
7.593e+01
6.903e+01
6.212e+01
5.522e+01
4.832e+01
4.142e+01
3.451e+01
2.761e+01
2.071e+01
1.381e+01
6.903e+00
0.000e+00
[m s^-1]
0
1.000
2.000 (m)
0.500
1.500

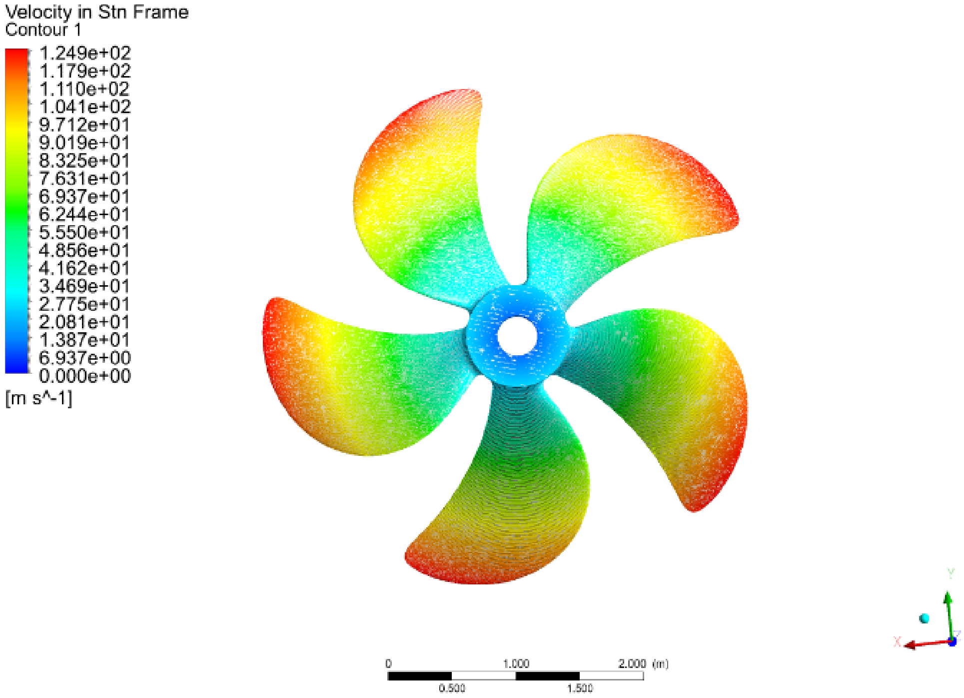
(d)


Velocity in Stn Frame
Contour 1
1.241e+02
1.172e+02
1.103e+02
1.034e+02
9.655e+01
8.965e+01
8.275e+01
7.586e+01
6.896e+01
6.207e+01
5.517e+01
4.827e+01
4.138e+01
3.448e+01
2.758e+01
2.069e+01
1.379e+01
6.896e+00
0.000e+00
[m s^-1]
0
1.000
2.000 (m)
0.500
1.500
Y
Z
X

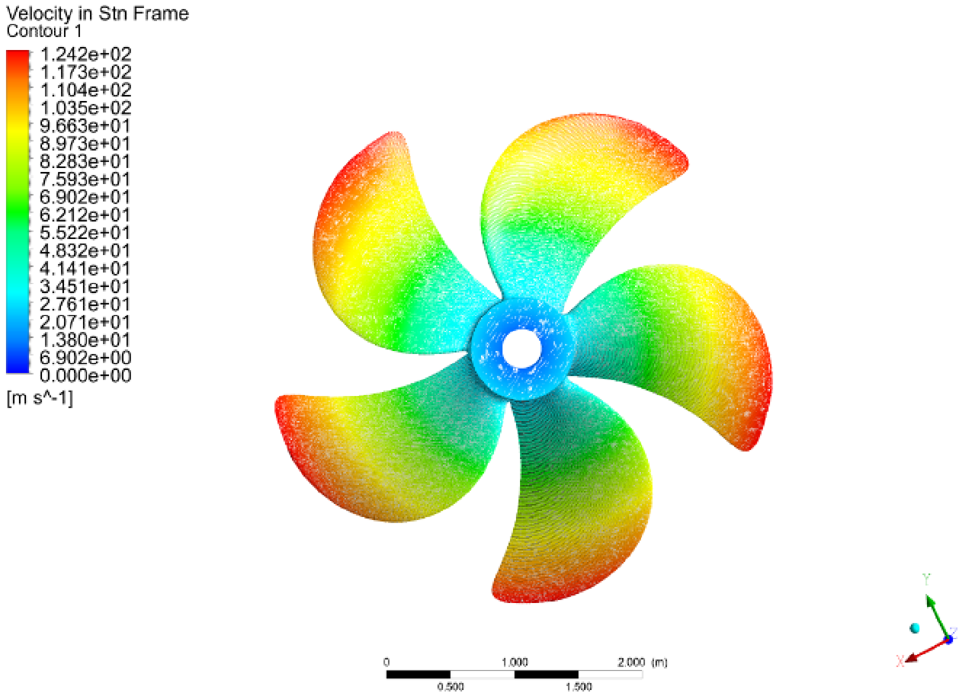
(e)


Velocity in Stn Frame
Contour 1
1.242e+02
1.173e+02
1.104e+02
1.035e+02
9.660e+01
8.970e+01
8.280e+01
7.590e+01
6.900e+01
6.210e+01
5.520e+01
4.830e+01
4.140e+01
3.450e+01
2.760e+01
2.070e+01
1.380e+01
6.900e+00
0.000e+00
[m s^-1]
0
1.500
3.000 (m)
0.750
2.250
Y
Z
X

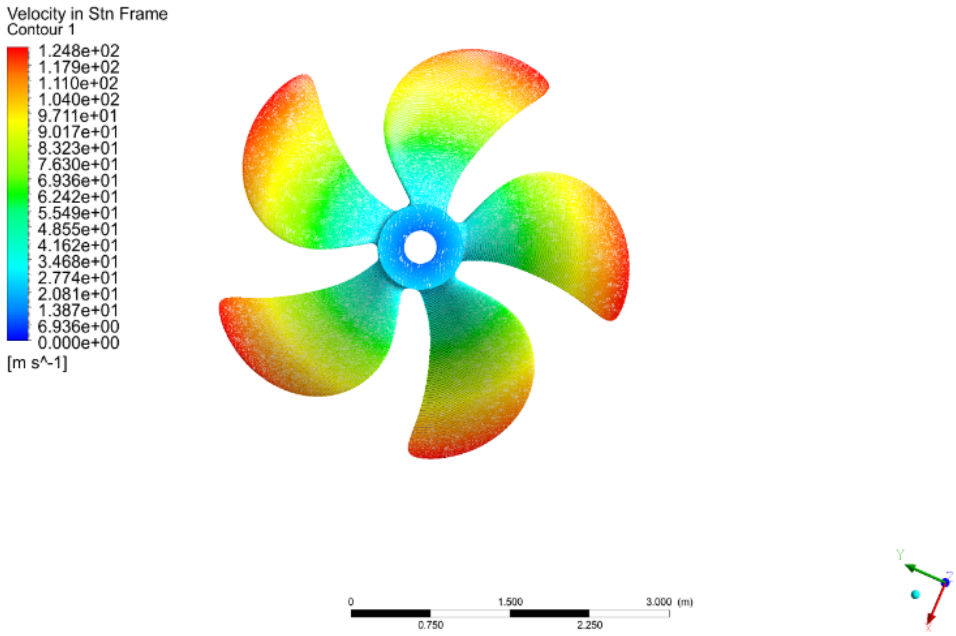
(f)


Velocity in Stn Frame
Contour 1
1.249e+02
1.179e+02
1.110e+02
1.041e+02
9.712e+01
9.019e+01
8.325e+01
7.631e+01
6.937e+01
6.244e+01
5.550e+01
4.856e+01
4.162e+01
3.469e+01
2.775e+01
2.081e+01
1.387e+01
6.937e+00
0.000e+00
[m s^-1]
0
1.000
2.000 (m)
0.500
1.500

(g)


Velocity in Stn Frame
Contour 1
1.242e+02
1.173e+02
1.104e+02
1.035e+02
9.663e+01
8.973e+01
8.283e+01
7.593e+01
6.902e+01
6.212e+01
5.522e+01
4.832e+01
4.141e+01
3.451e+01
2.761e+01
2.071e+01
1.380e+01
6.902e+00
0.000e+00
[m s^-1]
0
1.000
2.000 (m)
0.500
1.500

(h)



(i)

Fig. 5 Velocity distribution on nine test propeller

The thrust values were calculated from the simulation results. The nine of them were designed to obtain the maximum thrust produced by the propeller. The DOE reduces the number of iterations required by guiding towards the optimum result shown in Table V.

TABLE V
ORTHOGONAL ARRAY

SR NO.	No. of Blade	Camber Value	Pitch Delta	Chord at Hub	Thrust (N)	S/N Ratio
1	3	0.03	0.1	0.3	7004.33	81.72
2	3	0.04	0.2	0.35	6315.37	80.82
3	3	0.05	0.3	0.4	6715.23	81.35
4	4	0.03	0.2	0.4	7264.58	82.03
5	4	0.04	0.3	0.3	8065.03	82.94
6	4	0.05	0.1	0.35	5522.08	79.65
7	5	0.03	0.3	0.35	8790.30	83.69
8	5	0.04	0.1	0.4	11424.79	85.97
9	5	0.05	0.2	0.4	10268.46	85.04

Table V is used to set orthogonal array for all nine tests. This method helps to minimize the number of iterations required to simulate. After calculating thrust value, S/N ratio is calculated and followed by the sum of the performance that needs to be calculated to get the optimal combination. SN ratio for "Larger the better" is shown in (1):

$$\text{S/N ratio} = -10\log\left(\frac{1}{n}\sum_{1}^{n}\frac{1}{y^2}\right) \quad (1)$$

Thus, the SN ratio for the first case comes out to be;

$$\text{S/N ratio} = 81.72$$

Now S/N ratios are to be calculated for different levels and factor combinations. The following is the sum of the performance values shown in Table VI [11].

$$S/N\ ratio\ (1-1) = \frac{(81.72+80.82+81.35)}{3} = 81.30$$

$$S/N\ ratio\ (1-2) = \frac{(81.72+82.03+83.69)}{3} = 82.48$$

$$S/N\ ratio\ (1-3) = \frac{(81.72+79.65+85.97)}{3} = 82.45$$

$$S/N\ ratio\ (1-4) = \frac{(81.72+82.94+85.04)}{3} = 83.23$$

Similarly, other levels and factor combinations were calculated. Following is the sum of performance values which becomes S/N ratio after getting divided by the number of trails.

TABLE VI
S/N RATIOS FOR DIFFERENT LEVEL AND FACTOR COMBINATIONS

Factor	No. of Blade (A)	Camber Value (B)	Pitch Delta (C)	Chord at Hub (D)
Level 1	81.30	82.48	82.45	83.23
Level 2	81.54	83.24	83.30	81.39
Level 3	84.90	82.01	82.66	83.12

From the observations, it is clear that the propeller's optimum combination is A3 B2 C2 D1. The new value of optimized propeller is given in Table VII.

TABLE VII
NEW OPTIMIZE PROPELLER PARAMETER

Factor	Pitch Delta (C)
No. of Blade	5
Camber value	0.04

Pitch Delta	0.2
Chord at Hub	0.3

V. Conclusion

The present study's essential feature is implementing the DOE and CFD on marine propellers with different levels of characteristics and factors. The CAD modelling is completed with CAESES software. All tests were simulated with ANSYS fluent and the thrust value is determined from the post-processing.

Considering the hydrodynamic properties, the parametric study reveals that the five-bladed propeller's thrust is superior to those for a different number of the blade. The study also reveals that increasing the chamber value and pitch delta results in increasing the propeller's thrust value. The optimized propeller produces a higher thrust value than the other, and its value is 11924.50 N.

Optimization of the propeller design using the DOE has turned out quite successfully. All this led to being the vital foundation for the initial design of the marine propeller. Future studies should examine the relationship between propeller characteristic effects on cavitation of the propeller.

References

[1] R. Suryawanshi cfd used to design and optimization the marine propeller, Vol. 3, issue 6, IJSRED.
[2] A. Banik, M.R. Ullah, Computation of hydrodynamic characteristics of a marine propeller using induction factor method based on normal induced velocity, in: Procedia Eng., 2017. doi:10.1016/j.proeng.2017.08.125.
[3] E. Belhenniche Samir, A. Mohammed, I. Omar, Ç. Fahri, Effect of geometric configurations on hydrodynamic performance assessment of a marine propeller, Brodogradnja. (2016). doi:10.21278/brod67403.
[4] L.N. Loi, N.C. Cong, N. Van He, CFD results on hydrodynamic performances of a marine propeller, Tạp Chí Khoa Học và Công Nghệ Biển. (2019). doi:10.15625/1859-3097/19/3/13246.
[5] G. K. Saha, Md, H. I. Maruf, Md. R. Hasan, Marine propeller design using CFD tools.
[6] U. Hollenbach, O. Reinholz, Hydrodynamic Trends in Optimizing Propulsion, 2nd Int. Symp. Mar. Propulsors. (2011).
[7] G.-J. Zondervan, J. Holtrop, J. Windt, T. van Terwisga, On the Design and Analysis of Pre-Swirl Stators for Single and Twin Screw Ships, 2nd Int. Symp. Mar. Propulsors.
[8] J. K. Choi, H. T. Kim, An investigation on the effect of the wall treatments in RANS simulations of model and full-scale marine propeller flows, Int. J. Nav. Archit. Ocean Eng. (2014). doi.org/10.1016/j.ijnaoe.2020.12.001.
[9] K. Mizzi, Y.K. Demirel, C. Banks, O. Turan, P. Kaklis, M. Atlar, Design optimisation of Propeller Boss Cap Fins for enhanced propeller performance, Appl. Ocean Res. (2017). doi:10.1016/j.apor.2016.12.006.
[10] “Applied design of experiments and taguchi method” by K. Krishnaiah, K. Krishnaiah.
[11] S.S. Lim, T.W. Kim, D.M. Lee, C.G. Kang, S.Y. Kim, Parametric study of propeller boss cap fins for container ships, Int. J. Nav. Archit. Ocean Eng. (2014). doi:10.2478/IJNAOE-2013-0172.

Rohit Suryawanshi. Birth in Karad, (India) on 3rd of July 1997. I completed my undergraduate bachelor in engineering study from a lokmanya tilak college of engineering in mechanical in 2019, Navi Mumbai, India.

He works for more than two years for Schnell racing team. Later he joins shakshi engineering as a CAE Engineer as a full-time job. He published a paper on “cfd used to design and optimization the marine propeller” in IJSRED 2020. He also publishes a paper “Predicting the effect of different parameter on ship resistance using CFD” in IRJET 2020.

Mr Rohit has a membership of SAEINDIA. He also won grand for undergraduate thesis from the University of Mumbai.