AUPET Press
Technical Journal of Daukeyev University
Volume 1, Issue 2, 2021, pp. 69-75
ISSN 2788-5488

Study of Flow by Varying the Position of Maximum Chamber Thickness on Aerofoil

Jithendra Sai Raja Chada, Ganesh Nathipam, Sri Ram Deepak Akella, Phani Bhaskar Anusuri
Department of Mechanical Engineering, Pragati Engineering College, Surampalem, Kakinada 533001, India
Corresponding author: Chada Jithendra Sai Raja; csai0799@gmail.com

Received 22 April 2021; Received in revised form 06 May 2021; Accepted 20 August 2021; Published 27 August 2021
DOI: https://doi.org/10.52542/tjdu.1.2.69-75

Abstract. Aerofoil is a design with a curved exterior that gives the most commendable ratio of lift to drag. The aerofoil which allows for the flow at a margin range of angle has a more remarkable impact on the power spawning from the turbine. The conception and examination for the airfoil Structure have been implemented in the current work to raise the behavior of the flow with a commendatory condition by varying the position of maximum chamber thickness position thereby varying the angle of attack with 5 degrees interval from 0-25 degrees. The best model is studied with surface modifications to increase the flow separation and efficiency of lift.

Keywords: chamber thickness, angle of attack, CFD, lineation's, linear regression.

1. Introduction

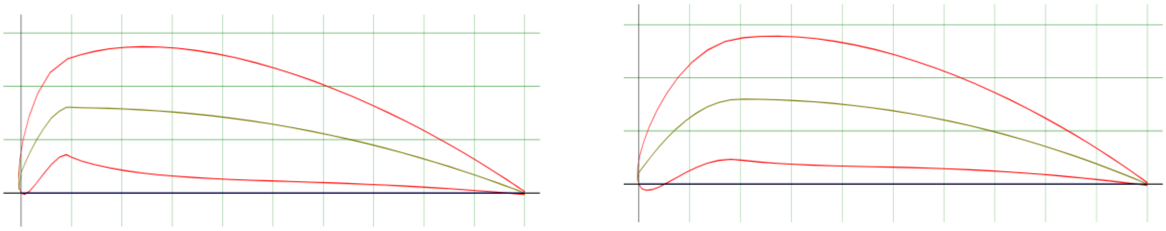
A steam turbine is an enlarged form of a heat engine, which propels most of its execution from the thermodynamics efficiency from various stages of steam generation. The steam turbine blades are aerofoil which is feather durable in nature, serves a better execution and efficient working and many other factors can be counted. The aerofoil passes through the fluid and produces aerodynamic forces which deliver a characteristic structure to work with higher efficiencies. This CFD analysis has delivered varying simulations for required outputs. As the aerofoil which is selected is of cambered type, the variation in the angle of attack leads to the variation in the velocity, pressure, and temperature contours of the considered aerofoil geometry. Jithendra Sai Raja Chada et al., have studied the flow on NACA 6412 of various dimple shapes. It is found that the aerofoil with circular dimples above and below the active surface has shown a better flow distribution [1]. Akhil Yuvaraj Manda et al., have studied the flow over NACA 6412, 7412, and 8412. The study concluded that the aerofoil 8412 has shown a better flow separation [2]. Rajat Veer et al., have conducted a simulated and experimental study on an aerofoil and concluded that the CFD study has similar results as experimental results [3]. Tulus et al., have studied the fluid flow around aerofoil with help of the Navier-Stokes equation and concluded that the Navier-Stokes equation results are close to experimental results [4]. Zubin Zaheer et al., have studied the flow on aerofoil and concluded that the lift increases as the difference in pressure between the active surface and lower surface [5]. A. Amit Kumar Saraf et al., have studied the flow separation on aerofoil with a bump. The Study stated that with help of bumps higher lift and reduced drag can be achieved [6]. Diksha Singh et al., have conducted an experimental study on the effect of dimples on aerofoil and concluded that semi-cylindrical dimples have given the best results [7]. Prasath M. S. et al., have studied the effect of dimples on aerofoil and stated that the dimples on the active surface reduce the drag on aerofoil [8]. Sandesh K. Rasal et al., have conducted the numerical analysis of lift & drag performance of NACA 0012 wind turbine aerofoil. It is concluded that the lift drag ratio has shown a higher value for an aerofoil with a dimple on the upper surface as compared to regular surface aerofoil [9].

2. Materials and Methods

The main intent of the work is to analyze the sensitivity of the flow along the surfaces of the aerofoil geometry and to calculate the velocity and pressure contours regarding the design of the geometry. The aerofoil geometry is various series depending upon the height of the upper chamber and lower chamber. As we consider, a constant chamber thickness of 8% and varying the maximum chamber position from 10% to 60%. As we have created a CAD part and performed flow simulations in solid works. We have made some amendments to the surface and created some dimples on it. The structure of the dimples is circular where the radius is 0.01 L. We have plotted the structure in solid works and created a sketch on it. We have done flow analysis on them. We have changed the angle of attack from 0° to 25° and performed flow analysis. The primary design is carried out in CATIA V5 using the plot points obtained from airfoiltools.com. The max chamber is fixed to 8% of chord length and the maximum thickness is fixed to 12%. The position of max chamber thickness is varied. The studied models are described in Table 1 and models are as shown in Figure 1.

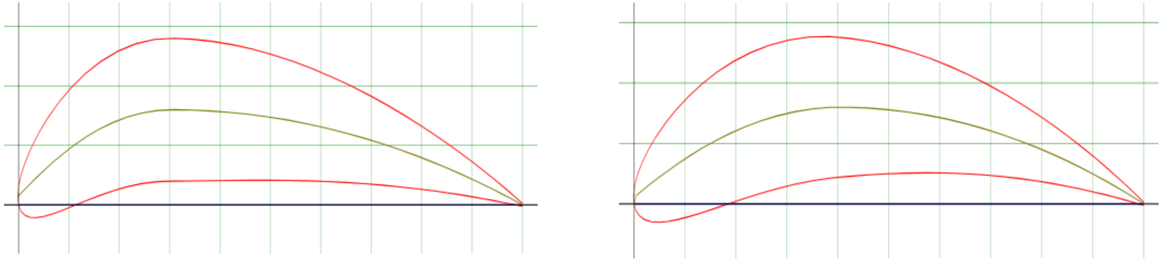
Table 1. Discrete models considered.

Model	Specification
M1	Max Chamber at 10% of L
M2	Max Chamber at 20% of L
M3	Max Chamber at 30% of L
M4	Max Chamber at 40% of L
M5	Max Chamber at 50% of L
M6	Max Chamber at 60% of L



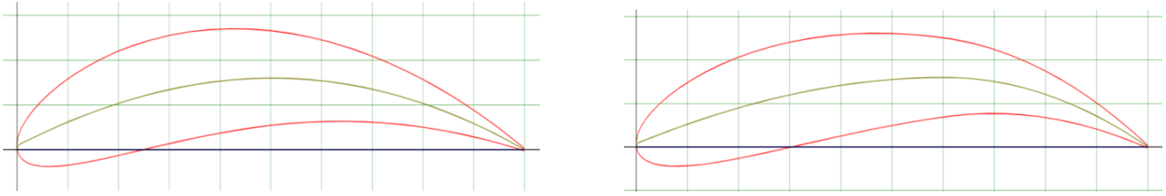
Aerofoil plot of M1

Aerofoil plot of M2



Aerofoil plot of M3

Aerofoil plot of M4



Aerofoil plot of M5

Aerofoil plot of M6

Figure 1. Reference plot of the aerofoil.

The finalized geometries are designed and evaluated in a predefined procedure as mentioned. The geometries are applied with principles such as Bernoulli’s equation and Navier – Stokes equation by varying the angle of attack with an interval of 5 degrees. The resulted model is studied with dimples on the active surface.

The evaluation concerning the calculation is termed as follows:

Bernoulli’s Equation

The application of principle between the principle of conservation of energy leads to a relation between the pressure, elevation and velocity of the flow in a fluid. This relation is called as the Bernoulli's Equation:

$$\Delta W = \Delta\left(K_E + P_E\right) \tag{1}$$

Work done equals force multiplied by distance:

$$W = Fd \tag{2}$$

We can plug in the formula that relates pressure and force, which gives us:

$$W = pAd \tag{3}$$

where A represents area.

Volume is derived by multiplying area and height (distance), thus:

$$W = pV \tag{4}$$

Work done is equal to:

$$\Delta W = p_1 V_1 - p_2 V_2 \tag{5}$$

Kinetic energy is the energy of mass in motion:

$$K_E = \frac{m\upsilon^2}{2} = \frac{\rho V \upsilon^2}{2} \tag{6}$$

where V represents volume.

Potential energy is dependent on height:

$$P_E = mgy = \rho V g y \tag{7}$$

where y represents height

Substituting gives:

$$p_1 V - p_2 V = \frac{\rho V \upsilon_2^2}{2} + \rho V g y_2 - \frac{\rho V \upsilon_1^2}{2} - \rho V g y_1$$

Divide by V:

$$p_1 - p_2 = \frac{\rho \upsilon_2^2}{2} + \rho g y_2 - \frac{\rho \upsilon_1^2}{2} - \rho g y_1$$

Rearranging the formula to put the terms that refer to the same point on the same side of the equation:

$$p_1 + \frac{1}{2}\rho\upsilon_1^2 + \rho g y_1 = p_2 + \frac{1}{2}\rho\upsilon_2^2 + \rho g y_2 \tag{8}$$

where y_1 and y_2 heights of inlet and outlet.

Navier – Stokes equation acts as a base principle for all the CFD flow modeling and evaluations. The Navier – Stokes equation acts as the conservation of momentum principle. The final equation is represented as mentioned:

$$\rho \frac{\delta \upsilon}{\delta t} = -\nabla p + \rho g + \mu \nabla^2 \cdot \upsilon \tag{9}$$

where ρ - is the fluid density; v – is the fluid flow velocity; p – is the fluid pressure; μ - is the fluid dynamic viscosity; ∇ - is the del operator.

3. Results and discussion

The intent of the work is to concept and evaluates optimum aerofoil geometry with efficient values in all the possible causes. The simulations and evaluations are carried out using the SOLIDWORKS software. The aerofoil geometry is first altered with the position of maximum chamber thickness. The appreciable geometry with arrangement at a fixed area was further evaluated for the extension of dimples over the active surface. The optimum aerofoil with further evaluations was opted and altered in terms of angle of attack and thereby evaluated in terms of performance along with the coefficient of lift and drag. The conception and evaluation are termed and preceded as discussed. The concept models with maximum chamber thickness at various positions are evaluated for the outlet velocity lineation and outlet pressure lineation at an inlet velocity of 8 m/s, 8.5 m/s, 9 m/s, 9.5 m/s, and 10 m/s. The results obtained gave the optimum results for the model with maximum chamber thickness at 30% of chord length at an inlet velocity of 10 m/s.

Table 2. Lineation values for max chamber positioning.

% of L	velocity (m/s)	max v (m/s)	min v (m/s)	max p (Pa)	min p (Pa)
10	8	9.6016	0	101367.988	101272.886
	8.5	10.188	0	101373.567	101266.485
	9	10.729	0	101379.474	101259.711
	9.5	11.380	0	101385.648	101251.977
	10	11.957	0	101391.946	101244.029
20	8	9.887	0	101367.148	101272.956
	8.5	10.514	0	101372.422	101266.452
	9	11.030	0	101378.216	101259.106
	9.5	11.130	0	101378.22	101259.11
	10	11.54	0	101382.72	101242.28
30	8	9.412	0	101367.52	101287.81
	8.5	9.995	0	101373	101282.95
	9	10.589	0	101378.7	101277.83
	9.5	11.177	0	101384.86	101272.42
	10	11.771	0	101391.36	101260.61
40	8	9.158	0	101367.34	101290.12
	8.5	9.72	0	101372.75	101285.49
	9	10.302	0	101378.52	101280.78
	9.5	10.875	0	101384.72	101275.89
	10	11.442	0	101391.01	101270.31
50	8	9.058	0	101367.24	101289.91
	8.5	9.647	0	101372.6	101285.36
	9	10.195	0	101378.45	101280.5
	9.5	10.765	0	101384.55	101275.38
	10	11349	0	101390.88	101269.87
60	8	9.057	0	101367.01	101293.85
	8.5	9.622	0	101372.25	101289.58
	9	10.194	0	101378.04	101284.9
	9.5	10.773	0	101384.15	101280.55
	10	11.327	0	101390.49	101275.39

Table 2 results are the simulation values for the outlet velocity and outlet pressure lineation when the maximum chamber thickness is placed at different locations. The maximum values in terms of both lineation's can be observed at 30% of the chord length of the aerofoil. The aerofoil when modified, geometry contributes for the negative pressure to develop along the corner. This negative pressure contributes to the increase in the values of the velocity and pressure lineation values as shown in Figure 2 and Figure 3 respectively, thereby making it the optimum position to allow for the arrangement chamber thickness.

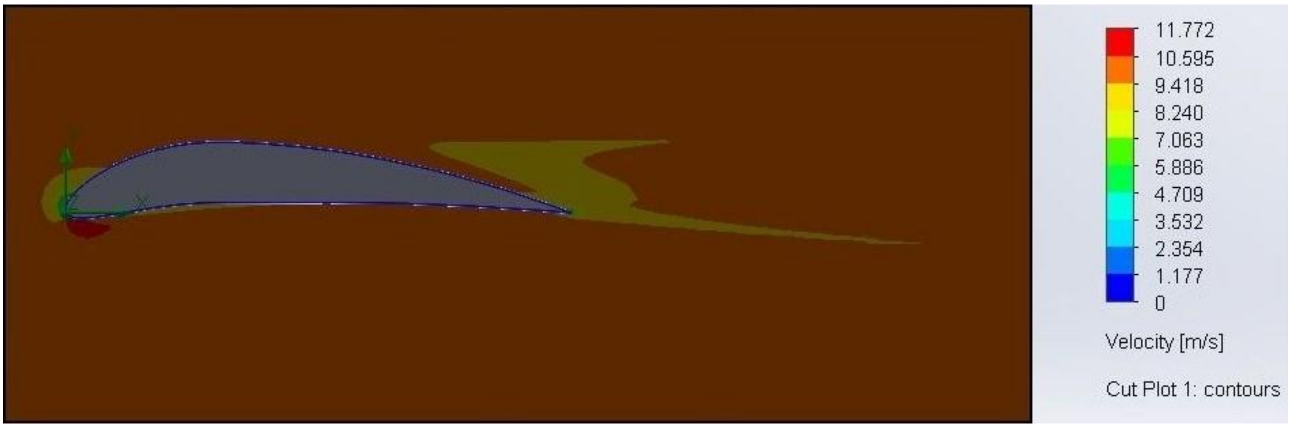


Figure 2. Velocity lineation for M3.

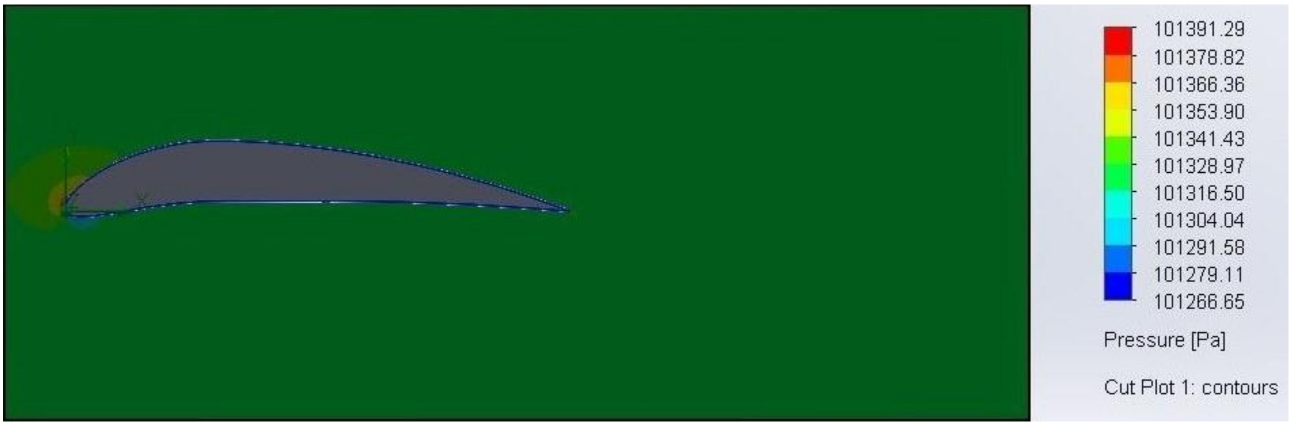


Figure 3. Pressure lineation for M3.

It is now further evaluated for the angle of attack to find an optimum angle of attack to place the aerofoil at a constrain. The M3 is evaluated by altering the angle of attack from 0^{o} – 25^{o} with a 5o interval. The results are shown in Table 3. The results plotted depict an appreciable deviation at an angle of 0^{o}.

Table 3. Variation of angle of attack to model 3.

AOA	max V	min V	max P	min P
0	11.771	0	101391.36	101260.61
5	11.441	0	101391.228	101268.089
10	11.26	0	101390.721	101277.18
15	11.189	0	101391.29	101276.873
20	11.075	0	101391.645	101278.936
25	11.0096	0	101388.449	101279.722

To find the error for the simulated data, regression analysis is done. Base on the R2 and adj R2 value, the error can be defined. The experimental values is close to 1 ie. 0.91, which indicate the regression data has good agreement with simulated data. The regression values are compared with simulated values in below Table. The final equation for regression is:

Velocity = - 0.02843 * angle + 11.65

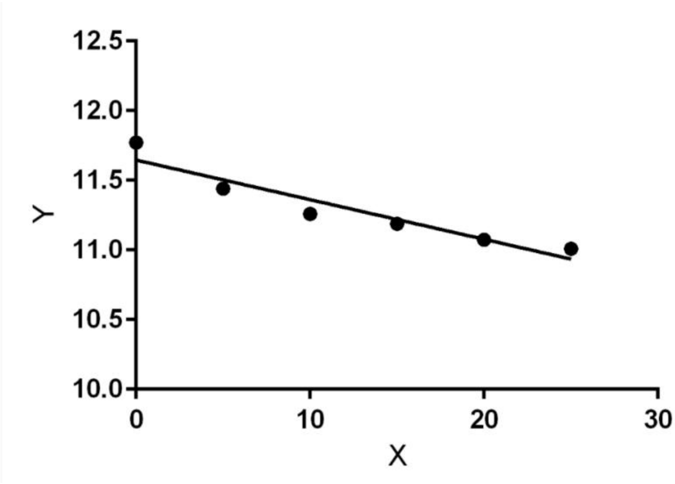


Figure 4. Regression plot between angle of attack and velocity.

Table 4. Comparison plot between the simulated data and the regression data.

Angle	Simulated result	Regression result
0	11.771	11.6464
5	11.441	11.5042
10	11.26	11.362
15	11.189	11.2198
20	11.075	11.0777
25	11.0096	10.9355

The M3 is applied with circular dimples for 30% of chord length is applied and studied the variation of flow with velocity and pressure lineation's are shown in Figure 5 and Figure 6. The velocity is varied from 12.0137946 m/s to 0 m/s. The maximum pressure is observed as 101395.273 Pa and the minimum pressure is 101239.258 Pa.

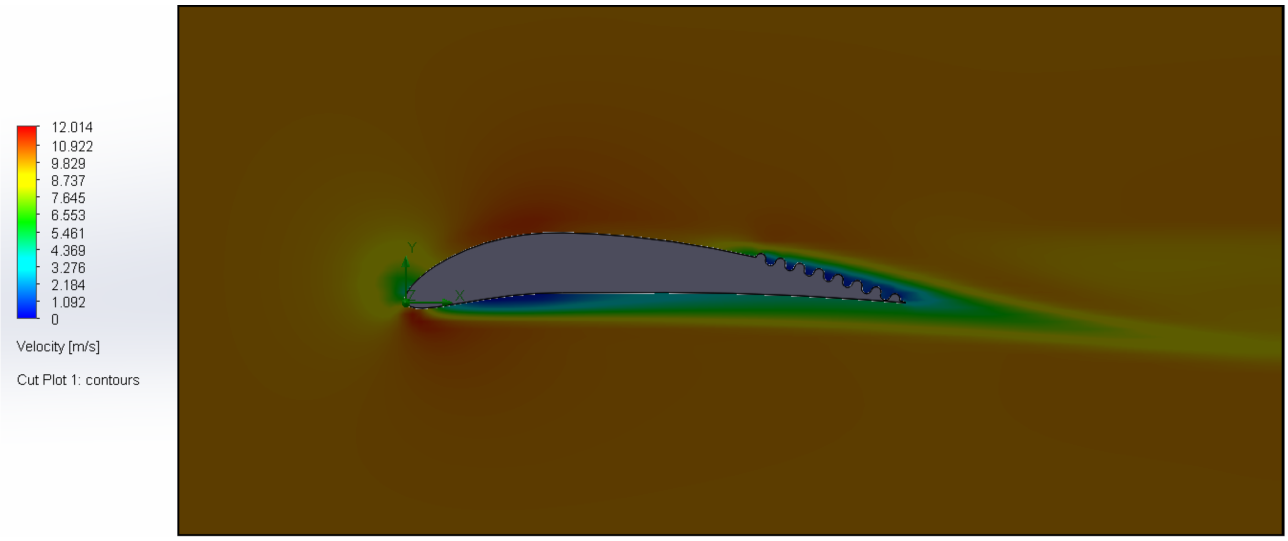


Figure 5. Velocity lineation on M3 with surface modifications.

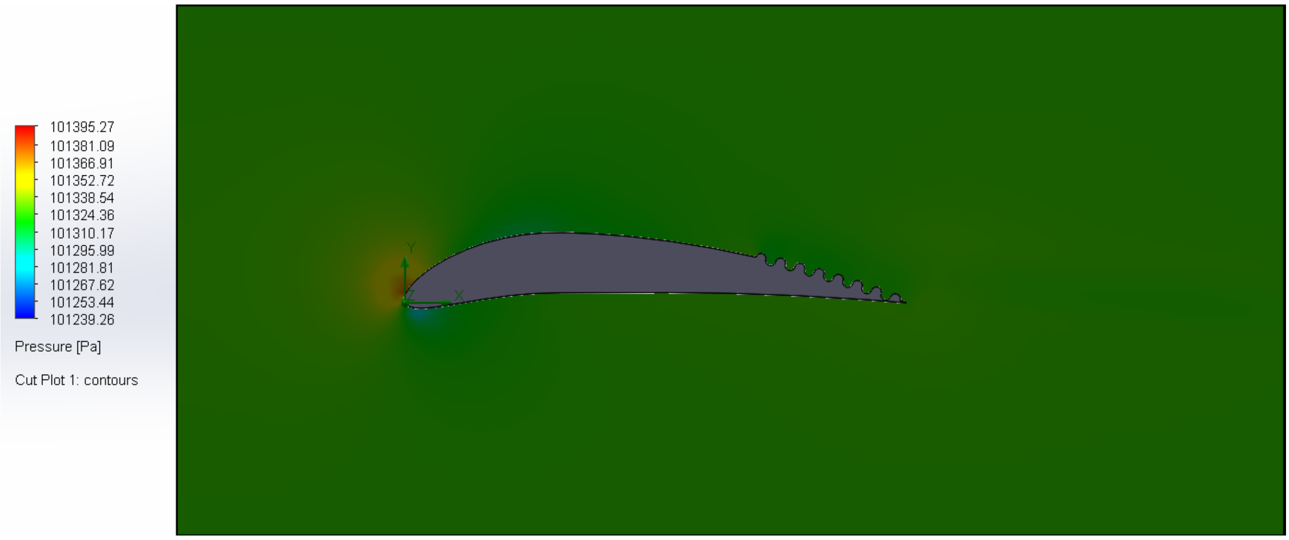


Figure 6. Pressure lineation on M3 with surface modifications.

4. Conclusions

The presented work targets to evaluate the flow behavior over the surface of an aerofoil with dimples spread at the rear end of the geometry. The aerofoil geometries are analyzed for the velocity lineation and pressure lineation. The geometries are modified by a change in the number of dimples over the surface i.e. 10% to 50% with an interval of 10%. The model M3 has depicted the highest outlet velocity of 11.771 m/s and the least pressure of 101260.61 Pa. The modified geometries are evaluated concerning the velocity and pressure lineation. The model has given the best results at 0 degrees inclination. The dimples on the active surface have further improved the flow separation as maximum velocity to 12.0137946 m/s and minimum pressure to 101239.258 Pa.

Reference

[1] Chada J. S. R., Satyanarayana K. V. V., Kumar G. V., Shaheen M., Bhaskar, A. P., (2020). Flow variation on NACA 6412 Aerofoil with surface modifications, I-manager's Journal on Mechanical Engineering, 10(3), 2020, 31-37.

[2] Manda Akhil Yuvaraj, Chada Jithendra Sai Raja, Surapaneni Sambhu Prasad, Geeri Satish, Flow behaviour on aerofoils using CFD, Journal of Mechanical Engineering, Automation and Control Systems, 1(1), 2020, 26-36.

[3] Rajat Veer, Kiran Shinde, Vipul Gaikwad, Pritam Sonawane, Yogesh Sonawane, Study and analyse airfoil section using CFD, International Journal of Engineering Research & Technology, 6(9), 2017, 47-51.

[4] Tulus C. Khairani, T. J. Marpaung, Computational analysis of fluid behaviour around airfoil with Navier-Stokes equation. Journal of Physics: Conference Series, 1376, 2019, 012003, 1-7.

[5] Zubin Zaheer, K. E. Reby Roy, Gopakumar S. Nair, Vineeth Ragipathi, U. V. Niranjan, CFD analysis of the performance of different airfoils in ground effect, Journal of Physics: Conference Series, 1355, 2019, 012006, 1-7.

[6] A. Amit Kumar Saraf, B. Mahendra Pratap Singh and C. Tej Singh Chouhanr, Study of flow separation on airfoil with bump, International Journal of Applied Engineering Research, 13(16), 2018, 12868-12872.

[7] Diksha Singh, Radheshyam H. Gajghat, Mrinal Kanti Manik, Experimental investigation to examine the effect of shape and size of dimples at suction surface of aerofoil, International journal of scientific & technology research, 8(12), 2019, 521-534.

[8] Prasath M. S., Irish Angelin. S., Effect of dimples on aircraft wing, Global Research and Development Journal for Engineering, 2(5), 2017, 234-242.

[9] Sandesh K. Rasal, Rohan R. Katwate, Numerical analysis of lift & drag performance of NACA0012 wind turbine aerofoil, International Research Journal of Engineering and Technology, 4(6), 2017, 2892-2896.