DESIGN OPTIMIZATION AND ANALYSIS OF AN AEROFOIL

Krishna Sharma[1], Utkarsh Dwivedi[2], Javed Alam[3], Abhishek Maurya[4], Vikram Kr. Saroj[5], Shah Fateh Azam[6], Md. Zahid Rayaz Khan[7]

[12345]UG Student, [6]Assistant Professor, [7]Head of Department, Department of Mechanical Engineering, KIPM College of Engineering and Technology, Gida, Gorakhpur, India

ABSTRACT

Aerofoil profile is very important in any aerodynamic system. While the shape of the aerofoil changes, its aerodynamic characteristics also change. Our study focuses primarily on changes in aerodynamic factors such as coefficient of lift (Cl) and coefficient of drag (Cd) due to minor variations in the coordinates of an aerofoil and angle of attack . We used standard symmetric aerofoil naca 0015 as reference because of its great performance in low wind speed and eight new aerofoil shapes introduced by changing chord thickness distance from leading edge of standard naca 0015 aerofoil without changing its maximum thickness in percent of chord . The aerodynamic airfoils of wind turbine blades have a crucial influence on aerodynamic efficiency of wind turbines. CFD (Computational Fluid Dynamics) analysis was carried out using ANSYS FLUENT 19.2 at various angles of attack from 0° to 12. Flow changes have been recorded for these aerofoil shapes and the results are arrived at for finding the best aerofoil that can be advisable to be used in wind turbines

Keyword:- *CFD, Co-Ordinate, Symmetrical Aerofoil, Coefficient of Lift, Coefficient of Drag*

---

1. INTRODUCTION

A wind turbine is acted upon by four aerodynamic forces; Thrust, Drag, Lift, and Weight. Wind turbines are able to produce torque due to the aerodynamic force produced when a fluid passes over the airfoil. An Airfoil is defined as the cross-section of a body that is placed in an airstream in order to produce an aerodynamic force in the most efficient manner possible[1]. If the pressure below the wing is higher than the pressure above the wing, there is a net force upwards and this upward force generates lift. In this project NACA 0015 aerofoil and its modified designs are used to analyze the factors like lift and drag coefficients, however, the term NACA is an abbreviation of (National Advisory Committee for Aeronautics), and the first digit in 0015 denotes the maximum camber, Cmax, as a percent of the chord. the second digit denotes the chordwise position of the maximum camber, XCmax, in tenths of the chord. the last two digits denote the maximum thickness of the airfoil section, t, as a percent of the chord.

1.1 PARAMETERS OF AEROFOIL

Leading edge - the forward most point on the airfoil (typically placed at the origin for convenience)
Trailing edge - the aft most point on the airfoil (typically placed on the x axis for convenience)
Chord line - a straight line between the leading and trailing edges (the x axis for our convention)
Mean camber line- a line midway between the upper and lower surfaces at each chord-wise position
Lift – A fluid flowing past the surface of a body exerts a force on it. Lift is the component of this force that is perpendicular to the oncoming flow direction. Lift conventionally acts in an upward direction in order to counter the force of gravity, but it can act in any direction at right angles to the flow.
Drag:-In fluid dynamics, drag (sometimes called air resistance, a type of friction, or fluid resistance) is a force acting opposite to the relative motion of any object moving with respect to a surrounding fluid. This can exist between two fluid layers (or surfaces) or a fluid and a solid surface.

Angle of attack::- Angle between the relative wind and chord line

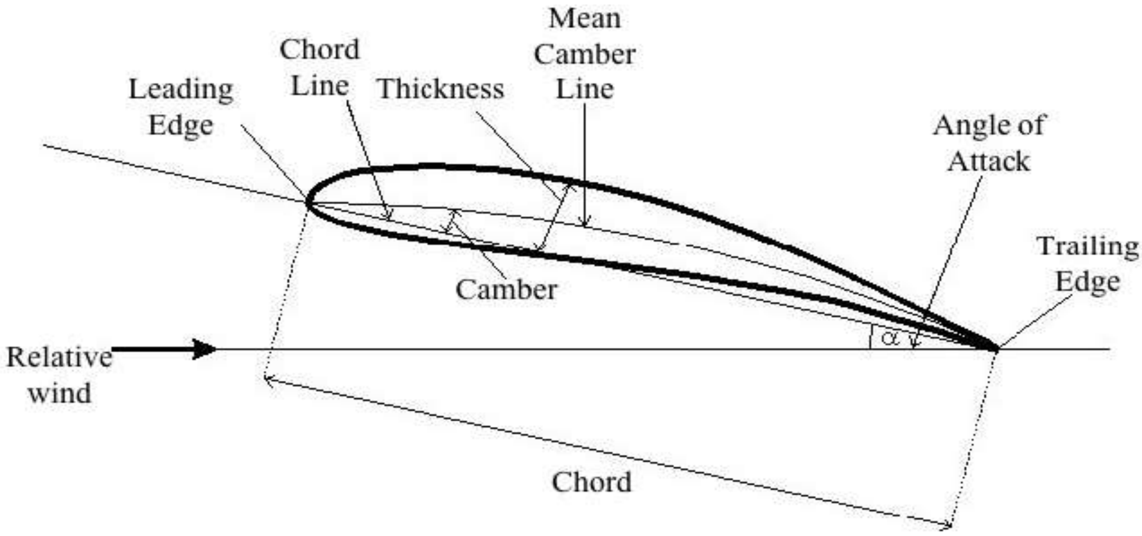

Figure 1. parameters of an aerofoil

1.2 KEY ELEMENT OF CFD

All the analyses were performed on Ansys 19.2 in this project. Ansys is an Engineering simulation software, it uses finite element analysis software to simulate computer models of structure or any machine component for analyzing the fluid flow or other attributes. Ansys is used to determine how a model will function with different specifications, without building a prototype or without conducting any kind of crash tests. And Due to the nonlinear behavior of the Navier-Stokes equations, solving a whole 3D turbulent flow model of a wind turbine rotor with the finest details in a time-dependent way is very difficult based on methods such as direct numerical simulation (DNS). Other options like large eddy simulation and detached eddy simulation methods are also applied in wind turbine aerodynamics by some researchers[2]. However, to be computationally cost-efficient, RANS equations are most generally wont to model the change of flow domain caused by turbulence around turbine blades. To obtain a reasonably accurate solution for wind turbine aerodynamics, three key elements are involved:
(1) A good mesh quality.
(2) An advanced turbulence model.
(3) An accurate solve scheme.

2. LITEREATURE REVIEW

Energy is essential to human civilization development. With the progress of economics and socialization, there's an expanding demand for renewable energy resources to secure energy supply, like solar energy, wind generation, tide and wave power, etc. As a clean natural resource, wind generation plays a more and more important role in modern life. according to Wikipedia Wind power accounts for nearly 10% of India's total installed power generation capacity and generated 62.03 TWh in the year 2018–19, which is nearly 4% of total electricity generation[3] A wind turbine converts kinetic energy into mechanical power through a rotor and then converts the mechanical power into electric power through a generator which is linked to the rotor with and without a gearbox. Various kind of wind turbines is designed to take advantage of wind power based on the principles of aerodynamics. Depending on the wind turbine rotor orientation, there are two types of wind turbines, horizontal axis wind turbine (HAWT) and vertical axis wind turbine (VAWT). Generally speaking, consistent with turbine capacity (size), modern wind turbines are often classified as small wind turbines (below 50kW), medium-size turbines (50kW~250kW), and enormous wind turbines (above 250kW). Several kinds of research have been carried out at different angles of attack and it is found that for naca 0015 drag coefficient has a minimum value at 70 angles of attack [4]. Its been also found from the study that the lift-to-drag ratio first increases monotonically from 00 angle of attack to 100 angle of attack and then due to an increase in form drag(Pressure Drag) lift-to-drag ratio decreases[5].

3. PROJECT AIM AND OBJECTIVES

The exploitation of small horizontal axis wind turbines provides a clean, prospective and viable option for the enhancement of energy supply. To reduce the risk in wind turbine development and improve the performance of the wind turbine systems, a better understanding of how these devices interact with the environment/winds is indispensable. This can be achieved via scaled-model laboratory experiments, full-scale field testing, or numerical modelling. It is clear that the benefits of numerical modelling are lower cost, lower risk and rapid design cycle, although it must be validated against measurements. An efficient approach for modelling the wind turbine blades is the computational fluid dynamics (CFD) method. The majority of the CFD approaches are based on the finite volume method, and discretisation is applied to the control volumes. It is a discretised computational analysis method for exploring the complex flows and wakes near the turbine blades. Moreover, the CFD approach provides a detailed quantitative analysis including blade surface pressure distributions, blade surface shear stress, and field pressure and streamlines. However, a particular CFD modelling is computationally expensive and has not been mature enough to become a design tool

4. NACA 0015 SYMMETRIC AEROFOIL

The reason to choose the NACA 0015 symmetric aerofoil is that it produces a more lift-to-drag ratio at low wind speed[6] and is the most common kind used for research purposes in most cases. Symmetrical aerofoil reduces the complexity of design and imparts easiness in the design modifications of the aerofoil. The center of pressure remains at a constant position as the upper and lower surfaces are identical in a symmetrical aerofoil. This reduces problems of Cp variations with varying angles of attack of airflow over the aerofoil. With changes in the positions of maximum thickness in percentages of chord and along the chord, the following profiles have been named as per their specification criteria[7]. It is to be noted that the amount of maximum thickness is not disturbed in this research content. The six modified profiles are shown below in Figure 2[8].

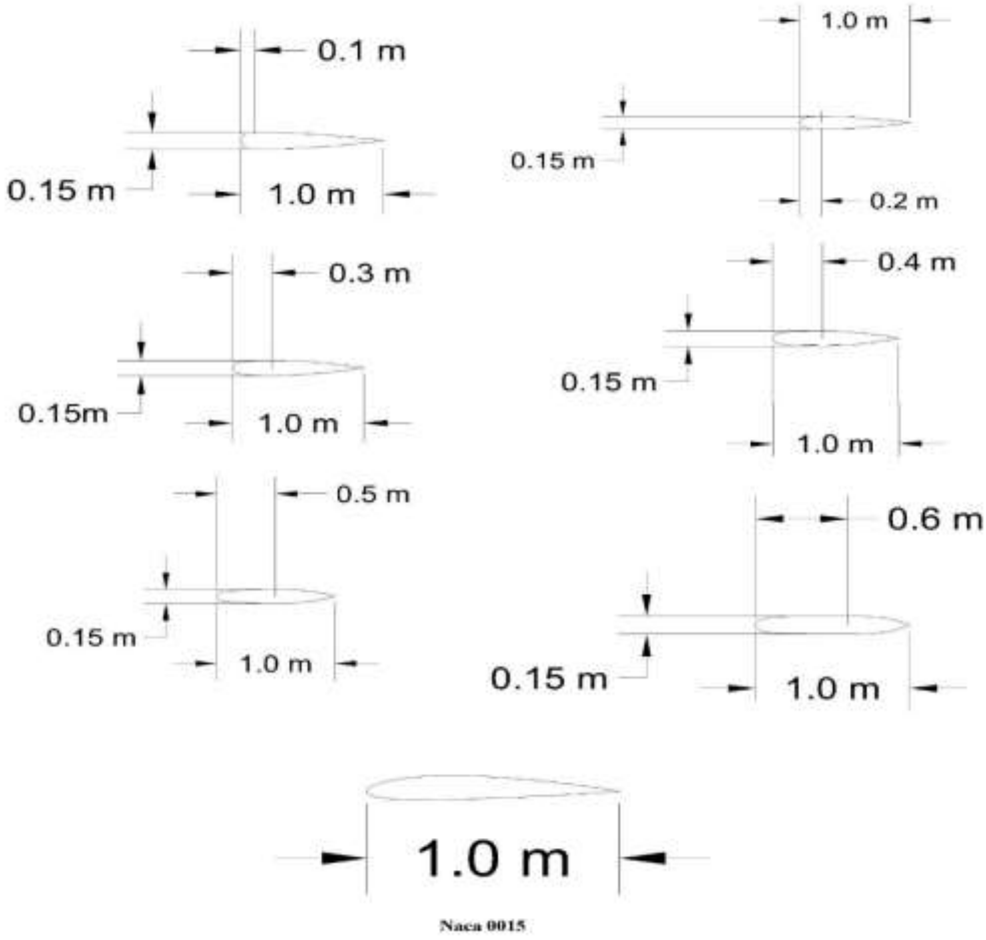

Figure 2. Six modified aerofoil design[8]

5. MODELLING AND MESHING

To model a turbine rotor using the CFD method, a precise 3D geometry of the turbine rotor is required in a digitized format, usually in a “computer-aided design” (CAD) format. A small turbine blade is usually twisted and tapered. The sectional airfoil of the blade may be a shape often with a little rounded leading edge, and a pointy edge or thin blunt edge[9]. A fine resolution of the boundary layer mesh is required to solve the boundary layer around the blade surfaces. To secure an accurate solution in the boundary flow, the dimensionless cell wall distance Y PLUS should be below or at least approximated to 1. And a large-enough flow domain is needed to avoid disturbances from the domain boundary surfaces, and a fine enough time step is preferable to generate a good result. However, a decent match between mesh refinement, mesh quality, domain size, and time step refinement is extremely important to provide a high-quality result.Final modelling and meshing are shown in the figure 3 and figure 4. . In meshing, element size is 15e-3 meter and airfoil model discretized into 277263 nodes and 1096404 elements.

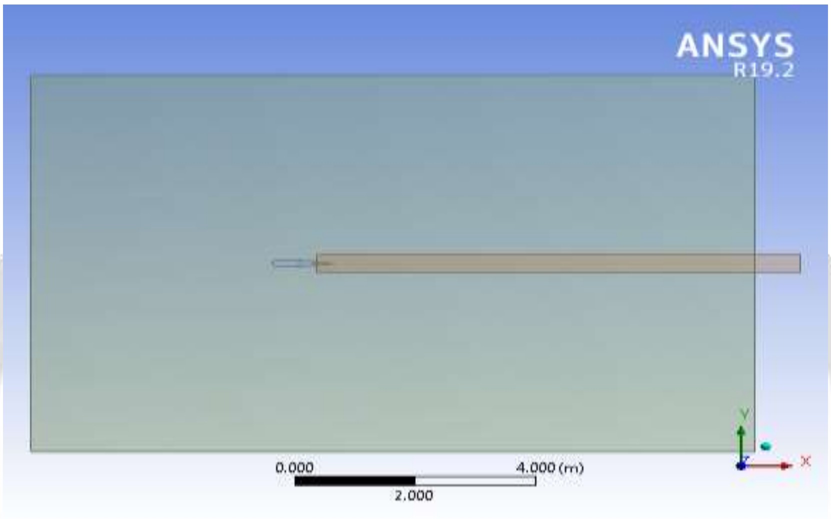

Figure 3. Front view of modelling

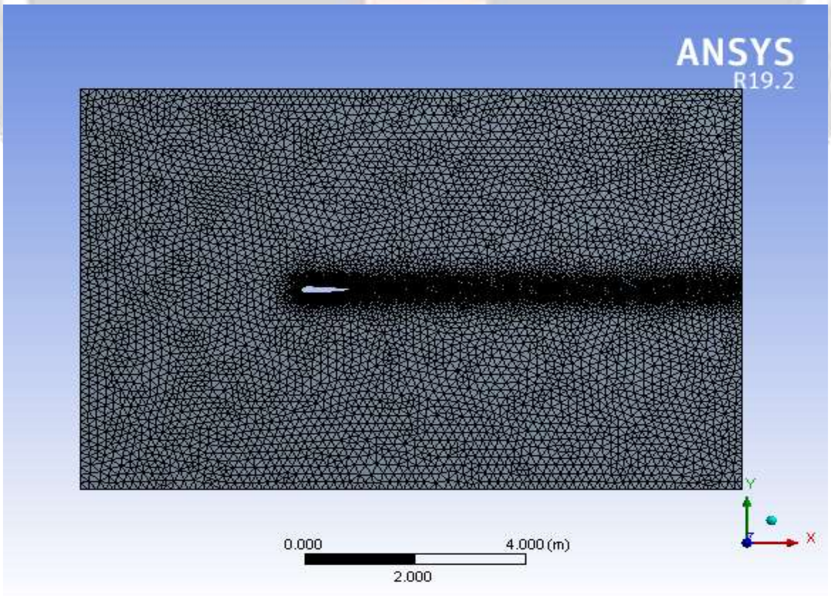

Figure 4. Final meshing(front view)

6. SIMULATION

The ANSYS-Fluent (version- 19.2) was used to analyze the flow analysis of flowing fluid over the airfoils. Imported CAD file of an airfoil in Workbench to generate a two-dimensional geometry of airfoil and also generated a twodimensional region around the geometry of airfoil as a flow region for flowing fluid and it's assumed that fluid flow in the z-direction is negligible. All the boundary conditions are applied with an option of Double Precision for all the airfoils in 3-D dimension. K-OMEGA SST MODEL (2 equation) flow equation has been chosen for the analysis of flowing fluid. The flowing fluid is considered an Air-Ideal gas and the flow of the flowing fluid is steady. The flow conditions are shown in the table below.

Table 1. Flow parameters set in fluent19.2

General	
Solver	Pressure based
Time	Steady
Models	
Viscous	k-omega (SST 2-equeation)
Material	
Fluid	Air
Fluid Density	1.225 kg/m3
Fluid Viscocity	1.7895e-05 kg/m-s
Boundary Condition	
Inlet velocity	10m/s
Outlet pressure(gauge)	0 Pa
Shear condition	No slip
Solution Methods	
Gradient	Least squares cell based
Pressure	Second order
Momentum	Second order upwind
Intialization	Hybrid Initialization

7. RESULT

7.1 COEFFICIENT OF LIFT

Lift is a mechanical force that's generated by a solid object passing through a fluid and this force opposes the weight of the flying object and holds it within the air. The lift is generated by the difference in velocity of the flying object and the fluid, around that flying object[10]. It takes no difference whether the thing is passing through the fluid or the fluid is flowing over an object.Coefficient of Lift is generated over the seven airfoils at different angle of attack to perform a comparative analysis of these airfoils on the basis of their predicted lift coefficient at angle of attack from 0^0 to 12^0. Table 2 and Chart1 represents the variation of lift coefficient at different angle of attack for these seven airfoils.

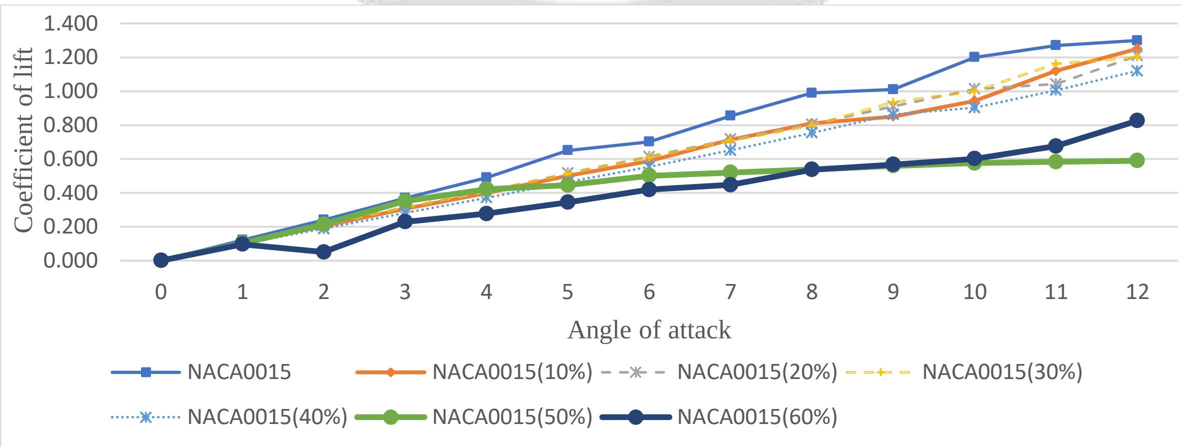

Chart 1. Coefficient of lift at sevral angle of attack

TABLE 2:- Coefficient of lift at sevral angle of attack for modified aerofoils

	AEROFOILS							
ANGLE OF ATTACK		NACA 0015	NACA 0015(10%)	NACA 0015(20%)	NACA 0015(30%)	NACA 0015(40%)	NACA 0015(50%)	NACA 0015(60%)
	0	3.0E-05	3.08E-05	5.780E-05	2.980E-05	6.580E-05	5.462E-05	2.910E-05
	1	1.2E-01	1.013E-01	1.060E-01	9.990E-02	9.767E-02	1.020E-01	9.677E-02
	2	2.4E-01	2.038E-01	2.082E-01	2.154E-01	1.889E-01	2.150E-01	5.070E-02
	3	3.7E-01	3.054E-01	3.142E-01	3.140E-01	2.819E-01	3.510E-01	2.290E-01
	4	4.9E-01	3.988E-01	4.060E-01	4.109E-01	3.709E-01	4.220E-01	2.770E-01
	5	6.5E-01	5.006E-01	5.161E-01	5.135E-01	4.640E-01	4.450E-01	3.446E-01
	6	7.0E-01	5.877E-01	6.133E-01	6.099E-01	5.530E-01	5.010E-01	4.190E-01
	7	8.6E-01	7.147E-01	7.163E-01	7.088E-01	6.510E-01	5.200E-01	4.472E-01
	8	9.9E-01	8.100E-01	8.045E-01	7.981E-01	7.540E-01	5.390E-01	5.370E-01
	9	1.0E+0	8.510E-01	9.098E-01	9.348E-01	8.631E-01	5.600E-01	5.677E-01
	10	1.2E+0	9.438E-01	1.013E+00	9.993E-01	9.030E-01	5.760E-01	6.012E-01
	11	1.3E+0	1.120E+00	1.043E+00	1.163E+00	1.006E+00	5.830E-01	6.757E-01
	12	1.3E+0	1.250E+00	1.210E+00	1.202E+00	1.120E+00	5.900E-01	8.264E-01
Note-10% means the maximum thickness of an aerofoil is at 10 percent of chord length from the leading edge								

7.2 COEFFICIENT OF DRAG

Coefficient of drag is generated over the seven airfoils at different angle of attack to perform a comparative analysis of these airfoils on the basis of their predicted drag coefficient at angle of attack from 0^0 to 12^0. Table3 and Chart 2 represents the variation of drag coefficient at different angle of attack for these seven airfoils.

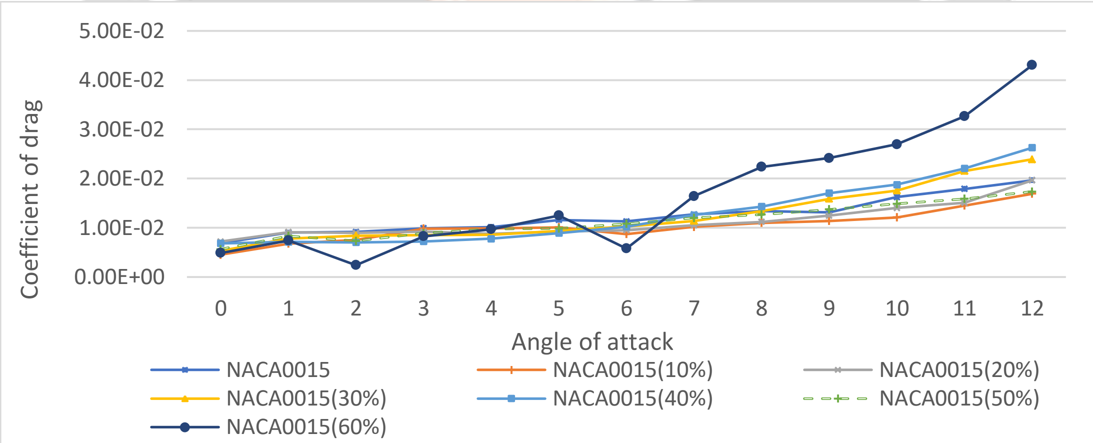

Chart 2. Coefficient of drag at sevral angle of attack

TABLE 3:- Coefficient of drag at sevral angle of attack for modified aerofoils

	AEROFOILS							
ANGLE OF ATTACK		NACA 0015	NACA 0015(10%)	NACA 0015(20%)	NACA 0015(30%)	NACA 0015(40%)	NACA 0015(50%)	NACA 0015(60%)
	0	6.9E-03	4.56E-03	7.20E-03	5.32E-03	6.84E-03	5.65E-03	4.89E-03
	1	9.0E-03	6.80E-03	9.14E-03	7.81E-03	7.13E-03	8.36E-03	7.44E-03
	2	9.2E-03	7.56E-03	8.98E-03	8.42E-03	7.05E-03	7.34E-03	2.43E-03
	3	9.9E-03	9.84E-03	9.21E-03	8.51E-03	7.25E-03	8.89E-03	8.30E-03
	4	1.0E-02	9.92E-03	8.77E-03	8.62E-03	7.81E-03	9.77E-03	9.72E-03
	5	1.2E-02	1.01E-02	9.27E-03	9.39E-03	8.94E-03	9.87E-03	1.25E-02
	6	1.1E-02	8.74E-03	9.51E-03	1.03E-02	1.02E-02	1.08E-02	5.82E-03
	7	1.3E-02	1.02E-02	1.05E-02	1.14E-02	1.26E-02	1.20E-02	1.64E-02
	8	1.3E-02	1.10E-02	1.12E-02	1.34E-02	1.43E-02	1.27E-02	2.24E-02
	9	1.3E-02	1.14E-02	1.25E-02	1.59E-02	1.71E-02	1.37E-02	2.42E-02
	10	1.6E-02	1.21E-02	1.40E-02	1.76E-02	1.88E-02	1.49E-02	2.70E-02
	11	1.8E-02	1.45E-02	1.51E-02	2.15E-02	2.21E-02	1.59E-02	3.26E-02
	12	2.0E-02	1.69E-02	1.96E-02	2.39E-02	2.63E-02	1.74E-02	4.30E-02

7.3 LIFT-TO-DRAG RATIO

The higher value of this ratio is the most desirable factor for any wind turbine. The value of the lift-to-drag ratio can be increased either by increasing the value of the lift coefficient or by decreasing the value of the drag coefficient It can be seen from the chart 2 and chart 3, that the value of lift coefficient and drag coefficient increases monotonically from the angle of attack 00 to 120, but the value of lift-to-drag ratio may or may not increases monotonically. Table 4 and chart 3 shows the variation of lift-to-drag ratio for different modified aerofoils at different angle of attack.

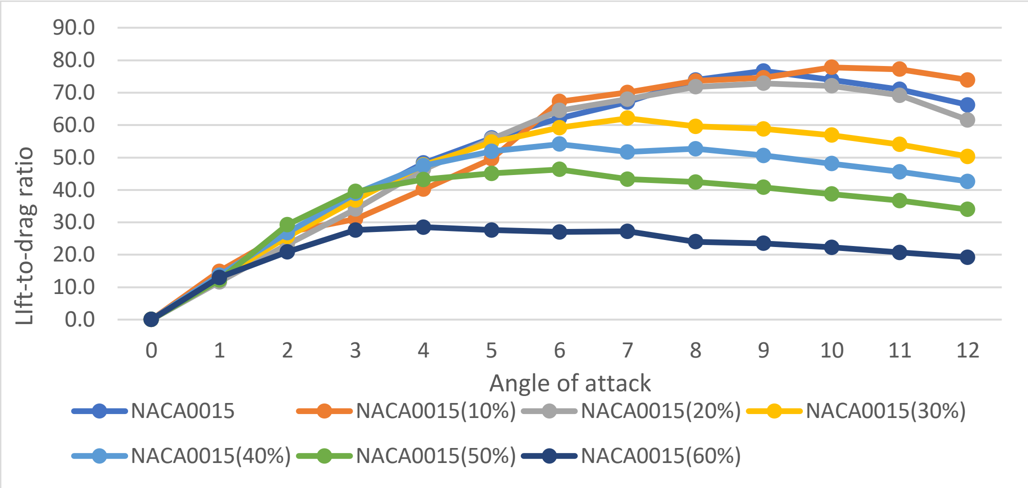

Chart 3. Lift-To-Drag Ratio at sevral angle of attack

TABLE 4:- Lift-to-Drag ratio at sevral angle of attack for modified aerofoils

	AEROFOILS							
ANGLE OF ATTACK		NACA 0015	NACA 0015(10%)	NACA 0015(20%)	NACA 0015(30%)	NACA 0015(40%)	NACA 0015(50%)	NACA 0015(60%)
	0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
	1	13.4	14.9	11.6	12.8	13.7	12.2	13.0
	2	26.2	27.0	23.2	25.6	26.8	29.3	20.9
	3	37.3	31.0	34.1	36.9	38.9	39.5	27.6
	4	48.3	40.2	46.3	47.7	47.5	43.2	28.5
	5	56.0	49.6	55.7	54.7	51.9	45.1	27.6
	6	62.1	67.2	64.5	59.2	54.1	46.3	27.0
	7	67.1	70.1	68.0	62.1	51.7	43.3	27.2
	8	73.9	73.6	71.8	59.6	52.7	42.4	24.0
	9	76.7	74.6	72.9	58.8	50.6	40.8	23.5
	10	73.9	77.8	72.1	56.9	48.1	38.7	22.3
	11	71.0	77.2	69.2	54.0	45.6	36.7	20.7
	12	66.2	73.9	61.6	50.3	42.6	34.0	19.2

8. CONCLUSION

The best aerofoil design for (L/D) ratio are shown as following:-

- At 1° AOA is Design 1 (10% of Chord);
- At 2° AOA is Design 5 (50% of Chord);
- At 3° AOA is Design 5 (50% of Chord);
- At 4° AOA is original NACA 0015.
- At 5° AOA is orignal NACA 0015.
- At 6° AOA is Design 1 (10% of Chord),
- At 7° AOA is Design 1 (10% of Chord),
- At 8° AOA is original NACA 0015,
- At 9° AOA is original NACA 0015,

- At 10° AOA is Design 1 (10% of Chord),
- At 11° AOA is Design 1 (10% of Chord),
- At 12° AOA is Design 1 (10% of Chord).

Among all these results, Design 1 with a maximum thickness at 10% of Chord length from the leading edge has been the best suitable aerofoil for use in turbines as it has a High (L/D) ratio at 10° of the angle of attack i.e. 77.8, and a better pressure gradient. Figure 5 and figure 6 show its pressure and velocity distribution around the aerofoil at 10° of the angle of attack respectively.

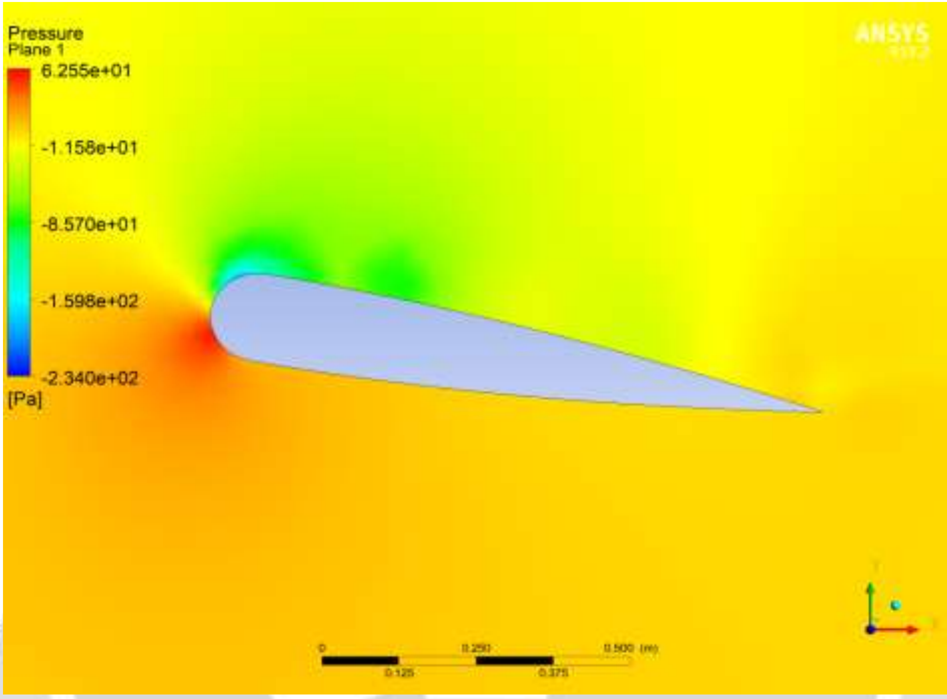

Figure 5. Pressure distribution of naca0015(10%) at 100 angle of attack

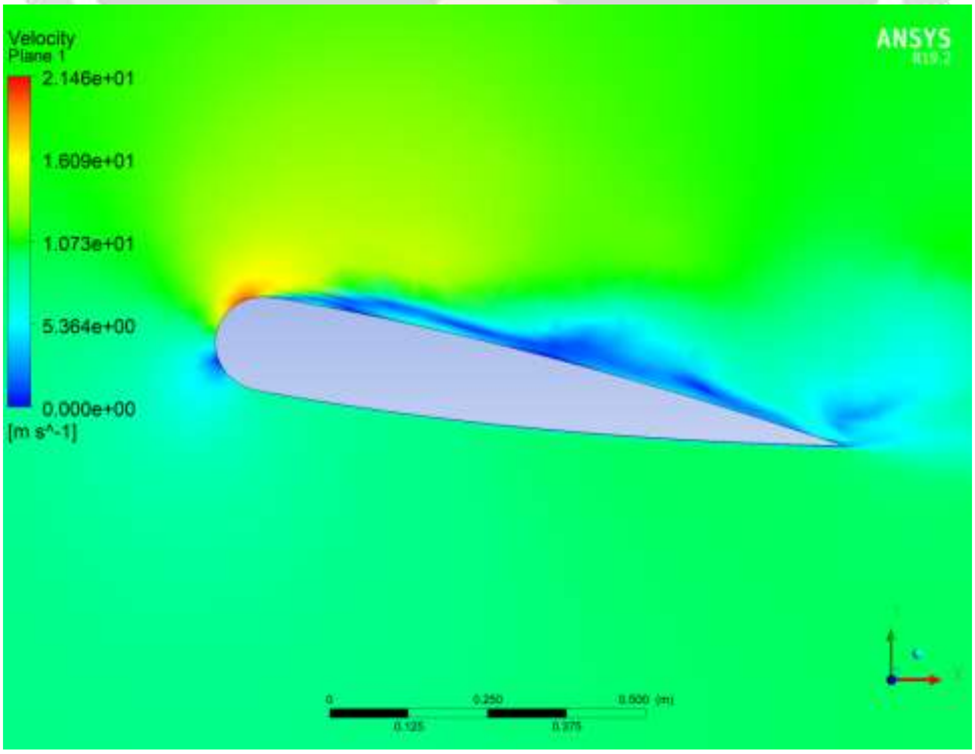

Figure 6. Velocity distribution of naca0015(10%) at 100 angle of attack

9.REFERENCES

[1] S.Kandwal and Dr. S. Singh, on "Computational Fluid Dynamics Study Of Fluid Flow And Aerodynamic Forces On An Airfoil", International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol. 1 Issue 7, September - 2012.

[2] Jiménez, A. 2009, "Analysis of a wind turbine wake using a LES method. Application to wake meandering".Euromech Colloquim on Wind Turbine Wakes Madrid.

[3] Wikimedia Foundation. (2022, April 28). Wind power in India. Wikipedia. Retrieved May 5, 2022, from https://en.wikipedia.org/wiki/Wind_power_in_India#cite_note-over-24

[4] Sharma, Ananth. S., S,Sudhakar., Swathijaya Kumar, & B S Anil Kumar. (2015). Investigation of pressure contours and velocity vectors of naca 0015 in comparison with optimized naca 0015 using gurney flap. International Journal of Mechanical And Production Engineering, Volume- 3, Issue-9.

[5] Kanimozhi, V. G. (2018). Analysis of Airfoil Flow Pattern using CFD. International Journal of Engineering Research & Technology(IJERT), Volume 6(Issue 14).

[6] Ameur, H., & Boukhadia, K. (2018, March). Numerical Study of the air flow over a NACA 0015 wind turbine airfoil. ResearchGate. Retrieved May 5, 2022, from https://www.researchgate.net/publication/345644694_Numerical_Study_of_the_Air_Flow_over_a_NACA_0015_Wind_Turbine_Airfoil

[7] Kondapalli, S. prasad. (2013, January). Aerofoil profile analysis and design optimisation - researchgate. ResearchGate. Retrieved May 6, 2022, from https://www.researchgate.net/publication/279186627_Aerofoil_Profile_Analysis_and_Design_Optimisation

[8] Dreese, J. (2018, January 24). DesignfoilVersion (6.04.0004). dreesecode. Retrieved from https://www.dreesecode.com/.

[9] Tang, Xinzi (2012, September). Aerodynamic Design And Analysis Of Small Horizontal Axis Wind Turbine Blades (thesis).

[10] Som, S. K., & Biswas, G. (2008). Introduction to fluid mechanics and Fluid Machines. Tata McGraw-Hill Publishing Company Limited.