

“CFD-Based Leading Edge Analysis of Aircraft Wing Aerofoil”

Dr.S .Ramasubramanian¹, Bose Vinodh², Uma Karthikav², Shreaiya.M³, J.Ragul³, Mohammed Haji Meerali.A³, Meghavarman.R³

¹HOD, Associate Professor, Department of Aviation, Vels Institute of science, Technology and Advanced Studies, (VISTAS), Chennai.

²Assistant Professor, Department of Aviation, Vels Institute of science, Technology and Advanced Studies, (VISTAS), Chennai.

³UG Students, Department of Aviation, Vels Institute of science, Technology and Advanced Studies, (VISTAS), Chennai.

-----***-----

Abstract:

The selection and design of aerofoils is the most important step in the aircraft design process. An aerofoil is the cross-sectional geometry of a wing that generates aerodynamic forces. When several aerofoils are piled together, they form a wing. The wing generates lift force, which counteracts the downward weight force. A complete examination of the shock, boundary layer, and other flow parameters must be undertaken before selecting any aerofoil for supersonic aircraft. Aerodynamic analysis is used to determine the vehicle's lift and drag coefficients, as well as estimate high- and low-pressure zones and separation points that affect vehicle performance. This analysis was done with CFD-Fluent, which is a fluid mechanics programme. For analysing and processing problems involving fluid flow, data structure and numerical analysis are used. The results of the aerodynamic analysis provide information on the many parameters and properties of the NACA 64-008A aerofoil. The analysis of a NACA 64 series supersonic aerofoil is the subject of this paper. The coefficients of lift and drag for three distinct leading edge radius at Mach 1.5 with a variable angle of attack are investigated. Pressure, velocity, and velocity stream line have been examined over the top and bottom surfaces of the NACA 64-008A aerofoil using Computational Fluid Dynamics.

Keywords: Supersonic Aerofoil; NACA 64-008A; viscous flow; Boundary Layer; CFD.

Introduction

An Aerofoil is a shape capable of producing lift with relatively high efficiency as it passes through the air. An aerofoil can have many cross sectional shapes. Aerofoil, An aerofoil is the cross-sectional shape of an item capable of generating significant lift through its motion through the air, such as a wing, a sail, or the blades of a propeller, rotor, or turbine. Even an item, such as a building or the deck of a bridge, blocks the wind, the object experiences drag as well as an aerodynamic force perpendicular to the wind. This does not imply that the object is an aerofoil, Aerofoils are highly efficient lifting structures that may provide much more lift with significantly less drag than similarly sized flat plates of the same area. Aerofoils are used in aircraft, propellers, rotor blades, wind turbines, and other aeronautical engineering applications. Different aerofoils are used to construct the aircraft wings. An airplane's wing has a special shape called an airfoil. The airfoil is shaped so that the air traveling over the top of the wing travels farther and faster than the air traveling below the wing. Thus, the faster moving air above the wing exerts less pressure

than the slower moving air below the wing. An airfoil is any part of an aircraft that is designed to produce lift. The wing is the primary airfoil but the propeller can also be an airfoil as well as the tail surfaces or sometimes even the fuselage itself. An airfoil has a leading edge, a trailing edge, a chord, and camber. The leading edge is the "front" of the airfoil—the portion that meets the air first. The trailing edge is the back of the airfoil—the place at which the airflow over the upper surface of the airfoil joins the airflow over the lower surface of the airfoil.

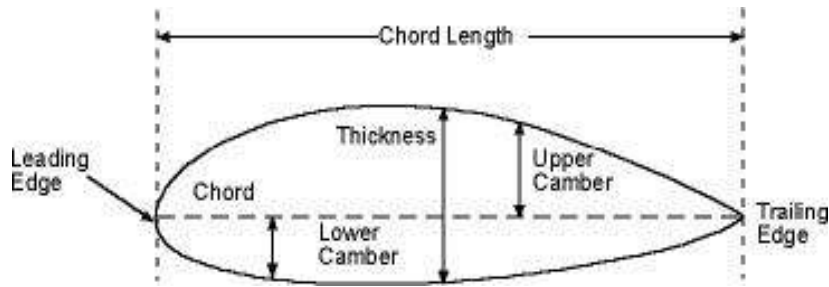


Figure : The elements of an airfoil

The chord of an airfoil is the imaginary straight line drawn through the airfoil from its leading edge to its trailing edge. The camber of an airfoil is the curve of its upper and lower surfaces. This curve is measured by how much it departs from the chord of the airfoil. Some airfoils have very little camber, i.e., the airfoil looks flat, while others have a higher degree of camber—the airfoil has more curve. The term upper chamber refers to the camber of the upper surface of the airfoil. The term lower camber refers to the camber of the lower surface of the airfoil. The camber of an airfoil affects its lift. The direction of the air that is flowing past an airfoil relative to the path of flight is called the relative wind. The relative wind is always parallel and opposite in direction to the path of flight.

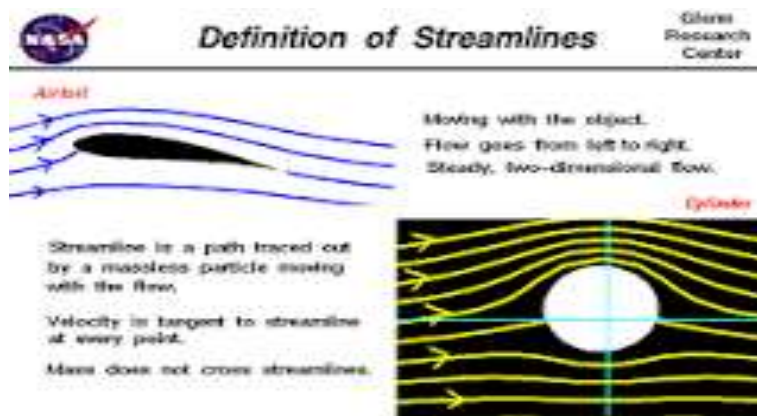


Figure No: Streamline of Aerofoil

Since no mass passes through the surface of the airfoil (or cylinder), the surface of the object is a streamline. Airplanes fly by moving through almost still air, yet we design and test them using wind tunnels, where the airplane model is fixed and the air is made to move past the model. The front of the wing (at the bottom) is called the leading edge; the back of the wing (at the top) is called the trailing edge. The distance from the leading edge to the trailing edge is called the chord, denoted by the symbol c . They are two types of aerofoils one is symmetrical and another one is non symmetrical aerofoil.

Since no mass passes through the surface of the airfoil (or cylinder), the surface of the object is a streamline. Airplanes fly by moving through almost still air, yet we design and test them using wind tunnels, where the airplane model is fixed and the air is made to move past the model. The front of the wing (at the bottom) is called the leading edge; the back of the wing (at the top) is called the trailing edge. The distance from the leading edge to the trailing edge is called the chord, denoted by the symbol c . They are two types of aerofoils one is symmetrical and another one is non symmetrical aerofoil.

Non symmetrical aerofoil, a cambered aerofoil is another name for it. The chord line is placed above with a large curvature, and the upper and lower surfaces are distinct. Different chord lines and chamber lines can be found in these. The lift-to-drag ratio and stall characteristics of non-symmetrical aerofoils are better, and usable lift is produced at zero angle of attack. The downsides are that they are not cost effective, and they produce unwanted torque

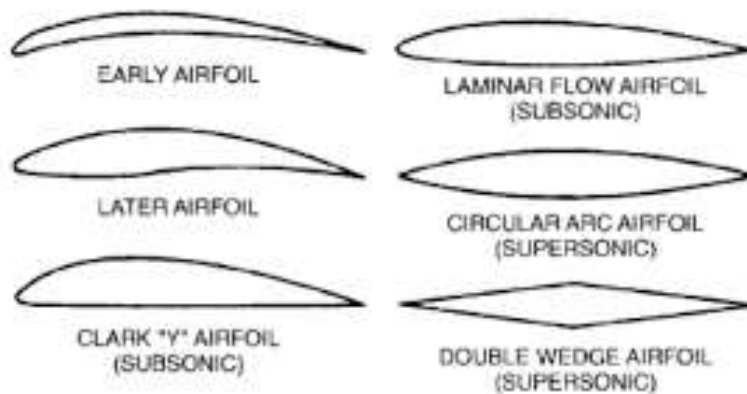


Figure No: Basic Shapes of Aerofoil

The angle at which relative wind meets an Aerofoil is known as the Angle of Attack. The angle created between the Chord of the aerofoil and the relative wind direction, or the vector expressing relative motion between the aeroplane and the atmosphere. The angle of attack is denoted by α and this alpha is a Greek word. The angle of attack of an aerofoil is principally responsible for its lift. The aerofoil deflects oncoming air at a proper angle, producing in a force on the aerofoil in the opposite direction of the deflection.

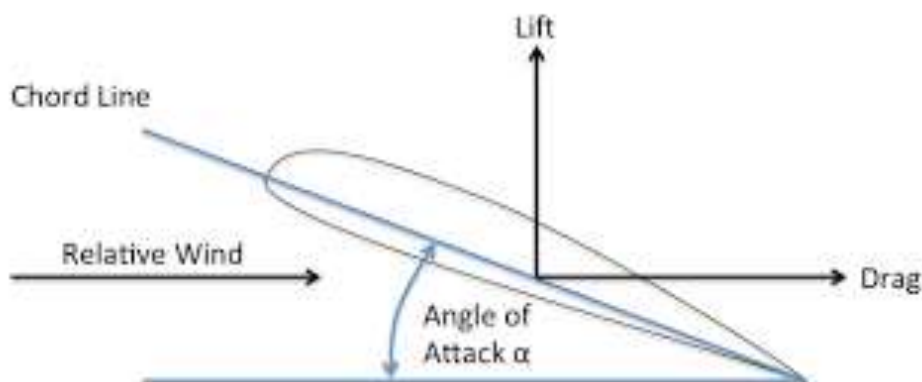


Figure No: Basic Shapes of Aerofoil

This force is referred to as aerodynamic force, and it has two components: lift and drag. The Lift, Thrust, Weight, and Drag are the four primary forces acting on the aeroplane. The aerofoil produces lift, and drag forces operate on it as a result the lift is the force that works perpendicularly upwards to forward motion, and the drag is the force that acts in the direction of the flow. When there is a pressure difference between the top and bottom surfaces of an aerofoil, lift is created. By calculating the well-known lift and drag characteristics of a symmetric double wedge aerofoil in supersonic flow, the benefits of spreadsheet computation are demonstrated.

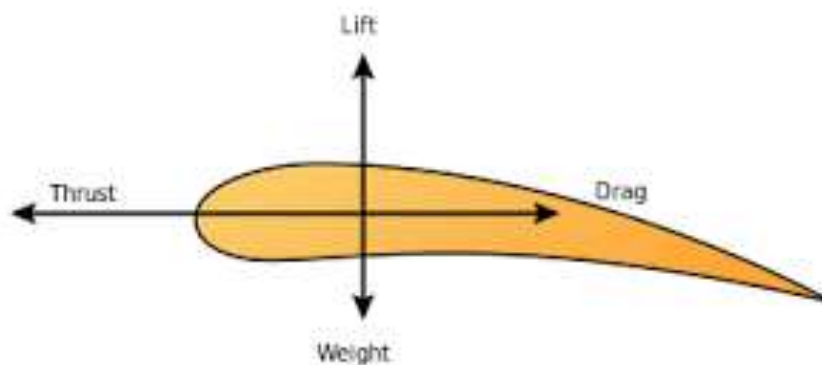


Figure No: Different Aerodynamic Forces

Techniques to Increase Lift in aerofoil:

- Increasing the angle of attack will increase the lift.
- Increasing the thickness will increase the lift.
- Increasing the area will increase the lift.
 - Increasing the altitude will decrease the lift.
 - Increasing the airspeed will increase the lift.
- Increasing the camber will increase the lift.
- A symmetric airfoil, or even a flat plate at angle of attack, will generate lift.
- Lift appears to be a very strong function of the airfoil camber.

When an aerofoil flows through a viscous fluid, a boundary layer forms around it, which is characterised by a low thickness, big velocity gradient. The magnitude of lift and drag produced by the aerofoil is changed by the flow regime changes. The classification of the aeroplane can be done based on the air velocity. A plane's speed is commonly expressed as a Mach number. This is the result of the object's speed being divided by the sound speed. Mach number is denoted by the letter M. It is characterised as a subsonic aircraft when the M is less than 0.8. Transonic aircraft travel at speeds between 0.8 and 1.2 metres per second, while supersonic aircraft travel at speeds

greater than 1.2 metres per second

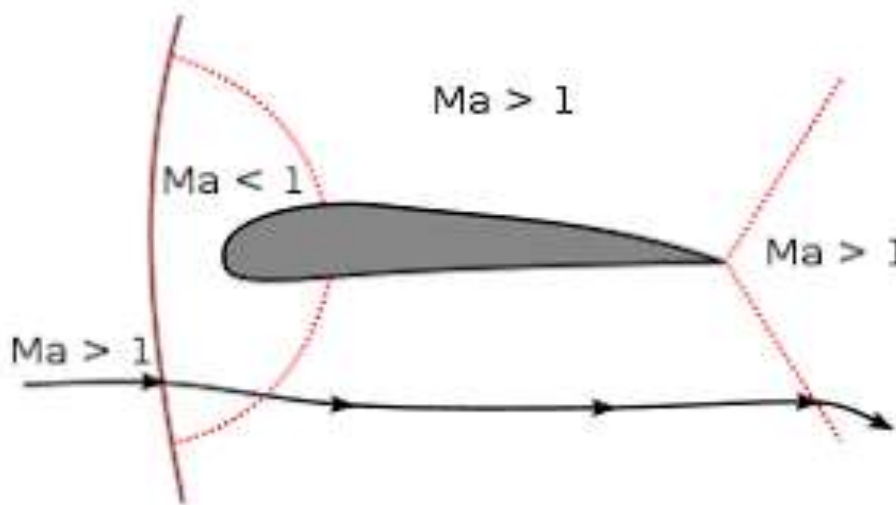


Figure No: Mach No in Aerofoil at different positions

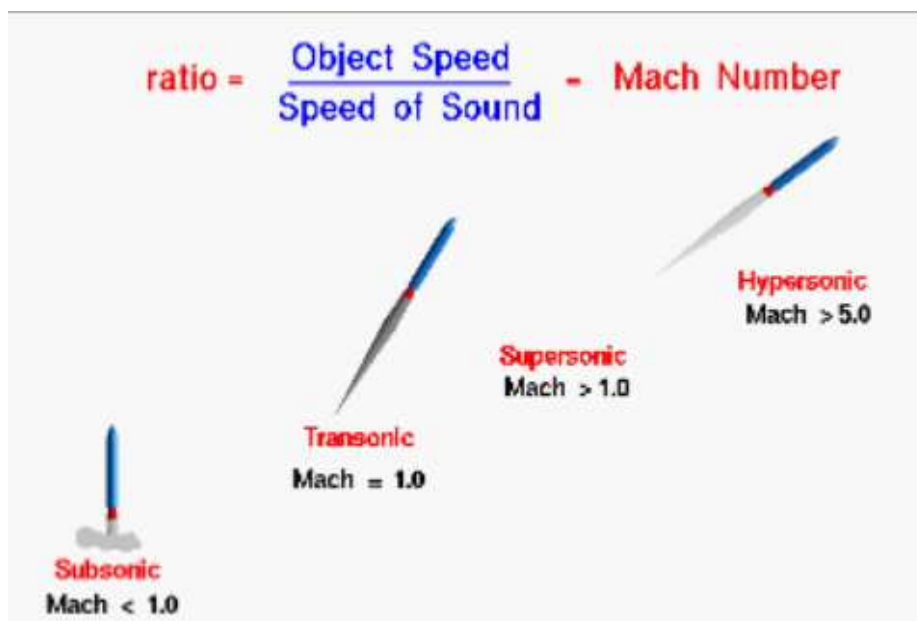


Figure No: Mach number and Aerodynamics Speed

By altering the shape of an aerofoil, it can be built for use at different speeds. Those optimised for subsonic, supersonic. All of them have a razor-sharp trailing edge. Hydrofoils are similar-functioning foils that use water as the operating fluid.

Subsonic. The upper surface of a typical subsonic wing is more curved than the bottom surface. The curved upper surface restricts airflow more than the flatter bottom surface, causing air above the wing to accelerate faster than air below. The lower the pressure, the faster the air speeds up, and this is called subsonic. This subsonic has the Mach number of 0.8-1.2.

Supersonic, the cross-section geometry of a supersonic aerofoil is designed to generate lift efficiently at supersonic speeds. When an aircraft must function consistently in the supersonic flying regime, such a design is essential. This supersonic aerofoil has the Mach number of 1.2-5 when we observe the supersonic aerofoil are thin.

Supersonic aircraft wings are thin to reduce drag, but a sweeping leading edge can provide lift, similar to a paper aeroplane. Vortex lift is the name for this sort of lift. Velocity is squared, as can be seen from the calculation above. This means that, unlike the Concorde, the wing does not need to be as efficient at high speeds to provide the needed lift. And there is an aerofoil called hypersonic aerofoil. Hypersonic aerofoil, if the Mach number is 5 or more than 5 and that aerofoil is said to a hypersonic aerofoil.

The majority of aerofoil shapes require a positive angle of attack to generate lift, however cambered aerofoils can generate lift at zero. Curved streamlines are created by the turning of the air near the aerofoil, resulting in lower pressure on one side and higher pressure on the other.

The primary goal of a supersonic aerofoil is to provide enough lift in the supersonic flow domain. These aerofoil have been designed to prevent the production of a disconnected bow shock. As a result, the aerofoil leading and trailing edges are kept razor-sharp. Aerofoil used in supersonic aircraft are often a narrow section made up of planes at an angle or arcs that are in opposition to one another. A shock is generated at a spot along the aerofoil when the local velocity of the fluid flow reaches sonic state.

CFD Analysis

Computational fluid dynamics (CFD) is a branch of [fluid mechanics](#) that uses [numerical analysis](#) and [data structures](#) to analyze and solve problems that involve [fluid flows](#). Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid ([liquids](#) and [gases](#)) with surfaces defined by [boundary conditions](#). With high-speed [supercomputers](#), better solutions can be achieved, and are often required to solve the largest and most complex problems. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as [transonic](#) or [turbulent](#) flows. Initial validation of such software is typically performed using experimental apparatus such as [wind tunnels](#). In addition, previously performed [analytical](#) or [empirical](#) analysis of a particular problem can be used for comparison. A final validation is often performed using full-scale testing, such as [flight tests](#).

CFD is applied to a wide range of research and engineering problems in many fields of study and industries, including [aerodynamics](#) and aerospace analysis, [hypersonics](#), [weather simulation](#), natural science and [environmental engineering](#), industrial system design and analysis, [biological engineering](#), fluid flows and [heat transfer](#), [engine](#) and [combustion](#) analysis, and visual effects for film and games.

Computational Fluid Dynamics (CFD) is the analysis of fluid flows using numerical solution methods. Using CFD, you are able to analyze complex problems involving fluid-fluid, fluid-solid or fluid-gas interaction. Engineering fields where CFD analyses are frequently used are for example aerodynamics and hydrodynamics, where quantities such as lift and drag or field properties as pressures and velocities are obtained. Fluid dynamics is involved with physical laws in the form of partial differential equations. Sophisticated CFD solvers transform these laws into algebraical equations and are able to efficiently solve these equations numerically.

CFD analyses have a great potential to save time in the design process and are therefore cheaper and faster compared to conventional testing for data acquisition. Furthermore, in real life tests a limited amount of quantities is measured at a time, while in a CFD analysis all desired quantities can be measured at once, and with a high resolution in space and time.

Because CFD analyses approximate a real physical solution, it should be noted that these CFD analyses cannot fully exclude physical testing procedures. For verification purposes tests should still be performed.

Objective of the work

The goal is to use Computational Fluid Dynamics to perform aerodynamic analysis over a supersonic aerofoil for three different leading edge radius and three different angle of attack (CFD). The aerofoil used in this study is NACA 64008. The leading edge radius is 4.8, 5, and 5.2, while the angle of attack is positive 5, 0, and negative 5 degrees during the analysis. ANSYS FLUENT is used to investigate the aerodynamic parameters for Mach 1.5.

MESHING AND EXPERIMENTATION

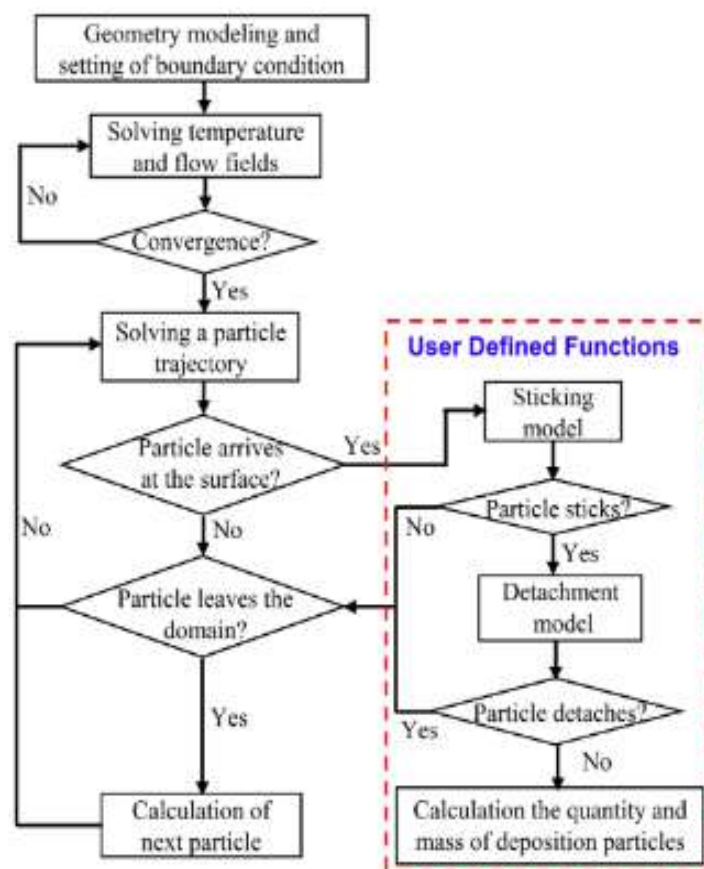


Figure: Flowchart of deposition computation

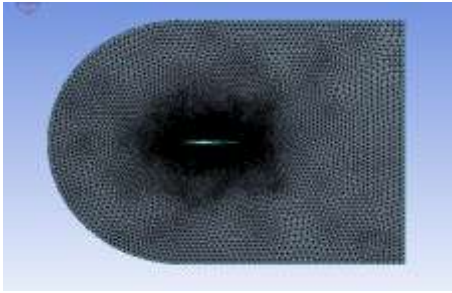


Figure: Complete mesh generation
Mesh Type: Triangular Mesh

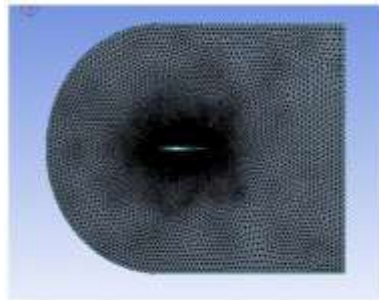


Figure : Leading Edge Radius 5 – 0 Degree AOA
of NACA64008 aerofoil

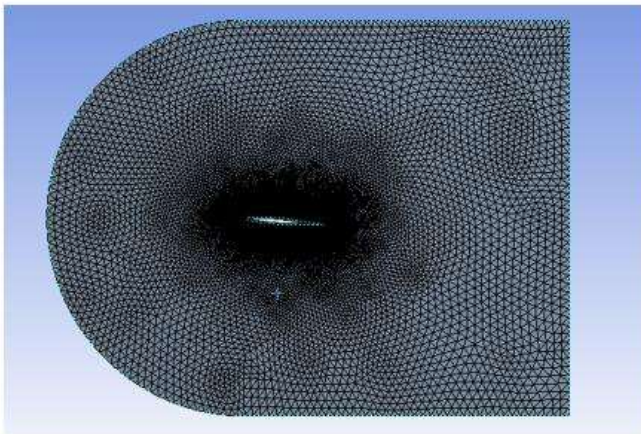


Figure : Leading Edge Radius 5 – 5 Degree AOA

Mesh Type: Triangular Mesh

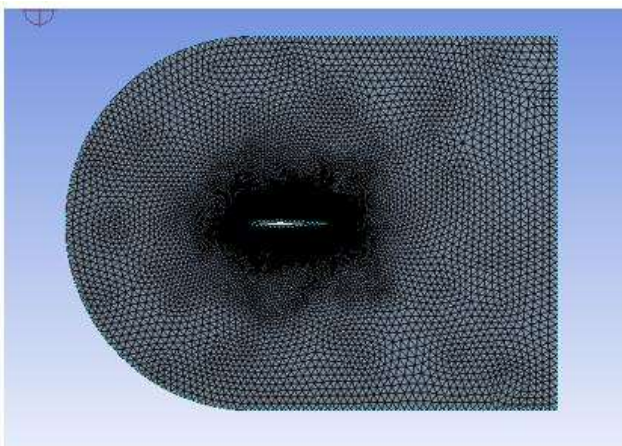
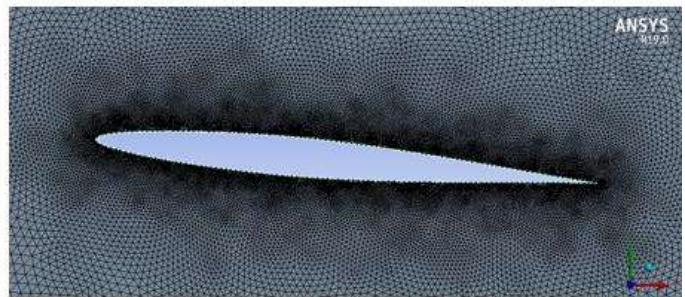
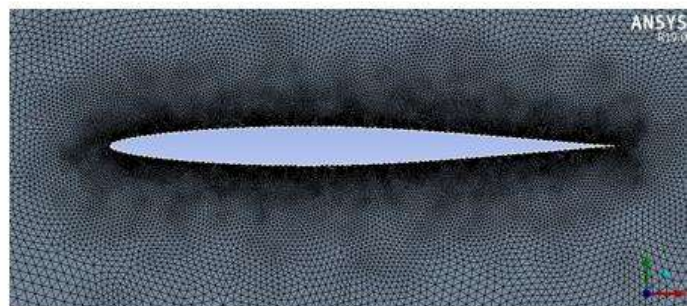


Figure No: Leading Edge Radius 4.8 – 5 Degree AOA

Mesh Type : Triangular Mesh



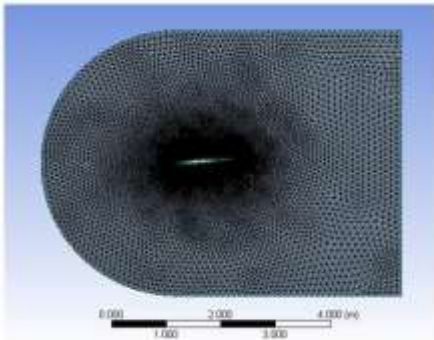


Figure: Leading Edge Radius
4.8– 5 degree AOA.

Mesh Type : Triangular Mesh



Figure No: Leading Edge Radius
5.2– 0 Degree

Mesh Type: Triangular Mesh

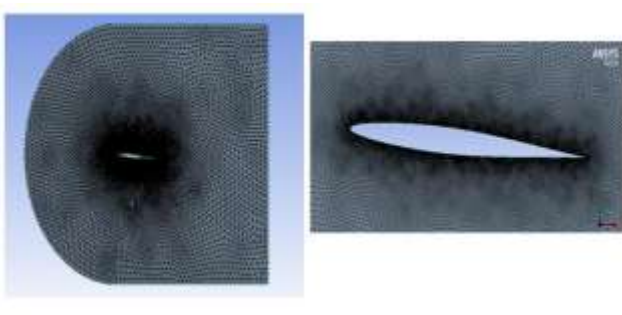


Figure No: Leading Edge Radius
5.2– 5 Degree AOA.

Mesh Type : Triangular Mesh

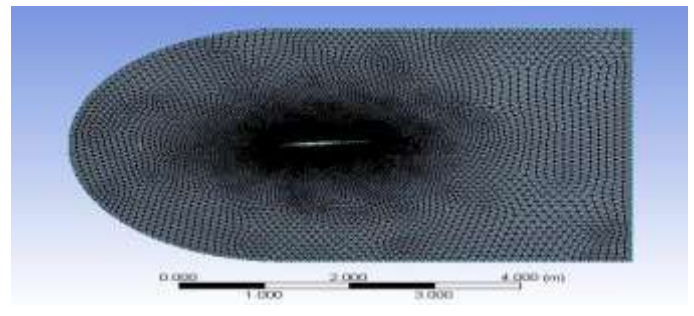


Figure No: Leading Edge
Radius 5.2– 5 Degree AOA

Mesh Type : Triangular Mesh

Computational fluid dynamics meshing adds a numerical grid to a fluid body and boundary. The precision of a computational fluid dynamics simulation is determined by meshing methods, which build groupings of grid points.

It is a triangular mesh grid. The NACA 64008 aerofoil has plotted. This is a triangular mesh because the aerofoil is in curved shape, for the curves shapes it gives better results than any other mesh and we have chosen this triangular mesh because Numerical diffusion is greater in unstructured triangular meshes than in structured meshes. This indicates that unstructured mesh solutions are more smeared out than structured mesh solutions with comparable element sizes.

In this case, the fluid used is air and the value of C_p is a constant i.e., 1006.43 J/Kg-K. Precision depends on the structure of the meshing part. The nodes were set at 30060 and the elements were at 58519. Precision again depends on the number of nodes and elements. Had the nodes and elements in the generated mesh be higher, the counter would have been accurate.

RESULT AND DISCUSSION

For leading edge radius 4.8, 5 and 5.2 for the three different angle of attack the lift, drag and coefficient of lift and coefficient of drag are calculated and the finding from the analysis are listed below in the following table

Table: Findings from the flow analysis.

Velocity	Leading edge radius	Angle of attack	Lift	Drag	Cd	Cl
517.5 m/s	4.8 (2% decrease)	0	16683.944	25101.953	0.15303146	0.10171194
		+5	45979.094	34502.897	0.21034334	0.28030679
		-5	-14095	36659	0.22349	-0.00859
	5	0	20377.148	24984.834	0.15231745	0.12422717
		+5	49052.583	33188.177	0.20232828	0.29904399
		-5	-42501	37537	0.22884	-0.25911
	5.2 (2% increase)	0	6018.7345	25472.816	0.15529238	0.036692591
		+5	50433.03	33827.209	0.20622408	0.30745974
		-5	-17714	36869	0.22477	-0.10799

For leading edge radius 4.8, 5 and 5.2 for the three different angle of attack the pressure and velocity contours are given below

1. Pressure counter

The pressure counter obtained for angles of attack 0°, 5° and -5° around the aerofoil is shown below. The findings from these pressure Counter for varied angle of attacks are further explained.

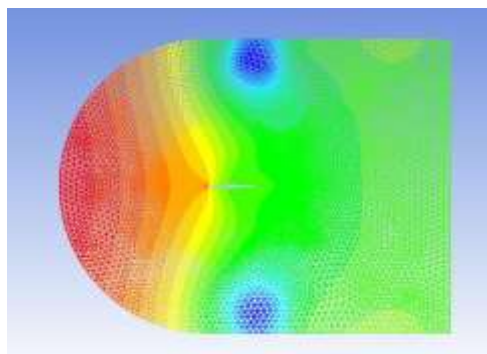


Figure: Leading Edge Radius 5 – 0 Degree AOA - Pressure Contour

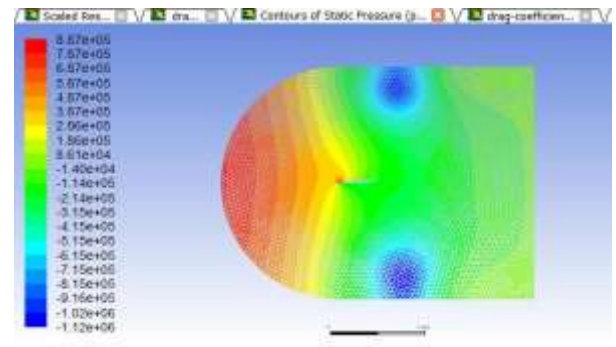
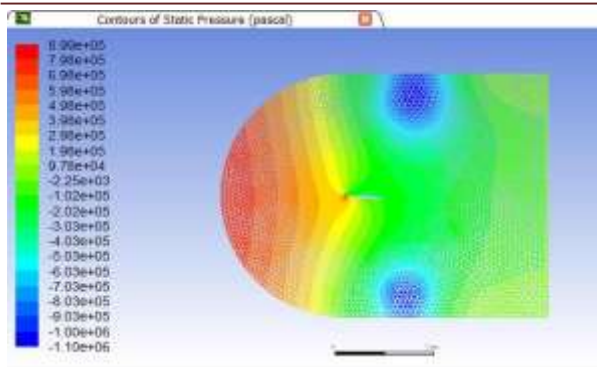


Figure No: Leading Edge Radius 5 – 0
Degree AOA - Pressure Contour

Figure: Leading Edge Radius 4.8– 0
Degree AOA - Pressure Contour

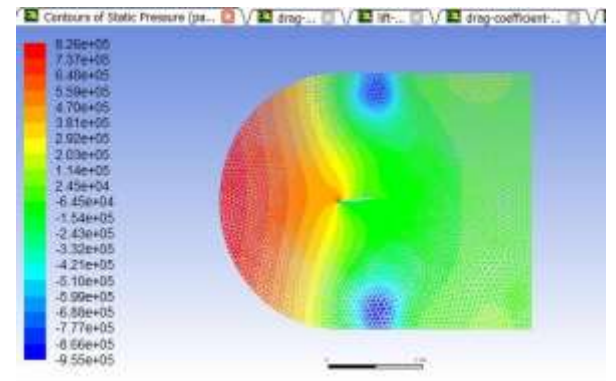
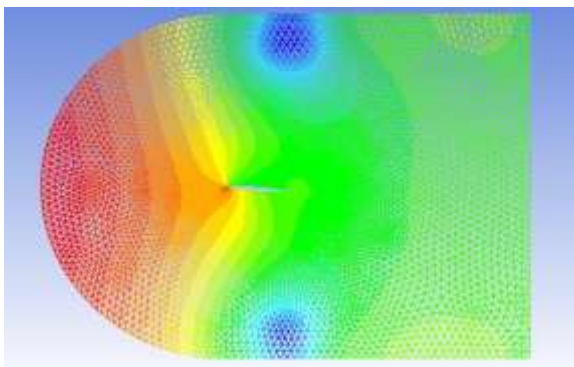


Figure: Leading Edge Radius 4.8– - 5
Degree AOA - Pressure Contour

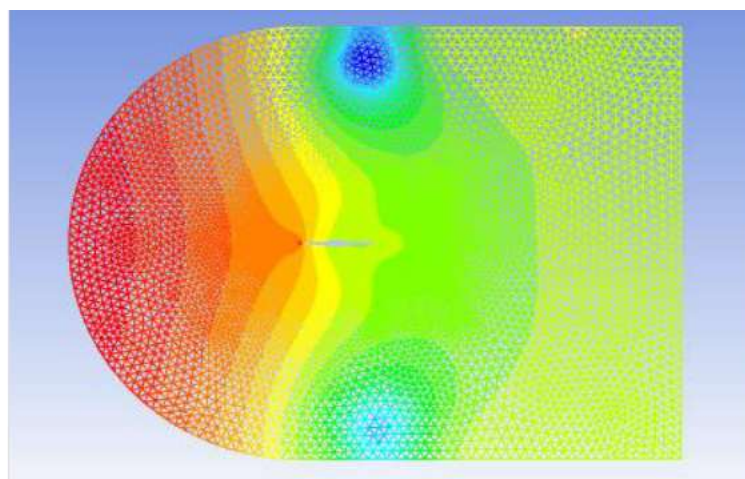


Figure: Leading Edge Radius 5.2– 0

Degree AOA - Pressure Contour

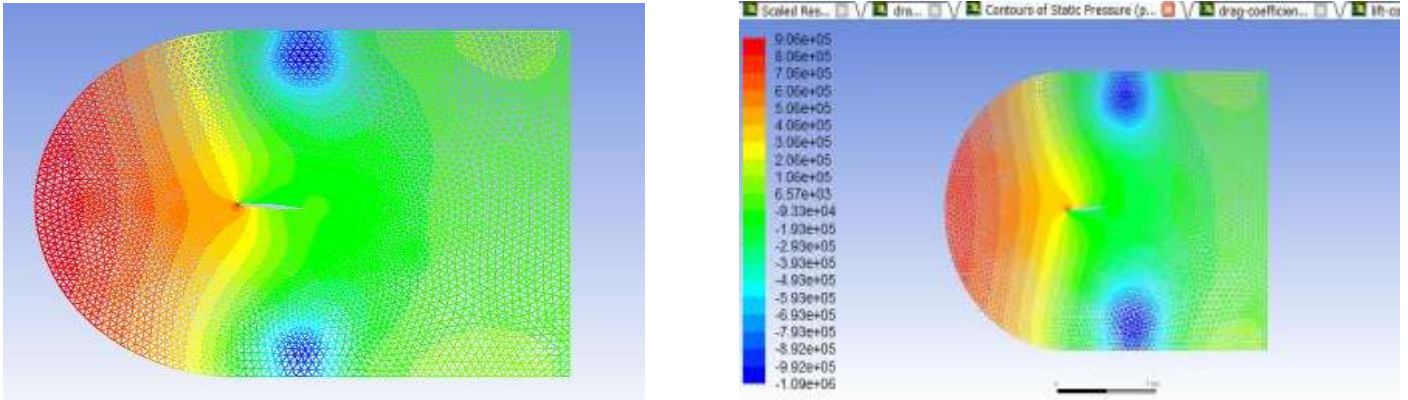


Figure: Leading Edge Radius 5.2– -5

Degree AOA - Pressure Contour

Discussion

For positive angle of attacks, a low pressure area is found above the aerofoil and a correspondingly high pressure area is found below the aerofoil, as shown in the pressure contours. Because of the difference in pressure, the lift force is activated. The lift force operates downwards with a negative angle of attack, but a very little positive lift force occurs for a 0o angle of attack. We can observe that as the angle of attack increases, so does the lift force. The presence of a high pressure area at the leading edge of the aerofoil is due to the same cause as the presence of a low velocity area at the leading edge

1. Velocity counter

The velocity counter obtained for angles of attack 0⁰,5⁰ and -5⁰ around the aerofoil is shown below. The findings from these velocity contours for varied angle of attacks are further explained.

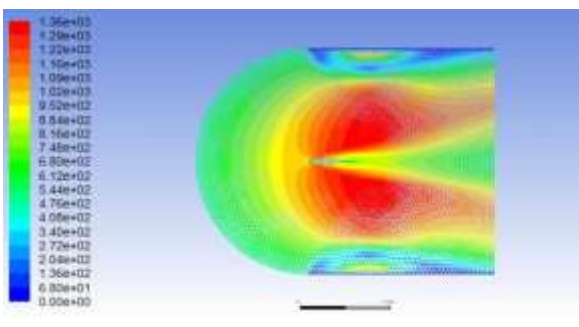


Figure: Leading Edge Radius 4.8– 0

Degree AOA-Velocity Contour

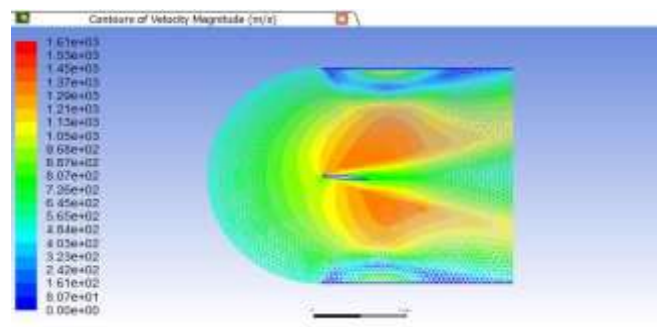


Figure: Leading Edge Radius 4.8– 5

Degree AOA - Velocity Contour

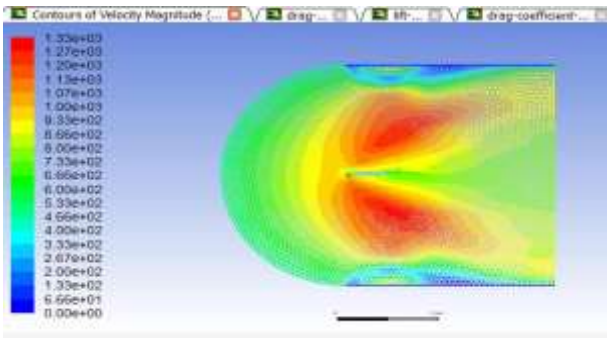


Figure: Leading Edge Radius 4.8 – 5 Degree AOA-Velocity Contour

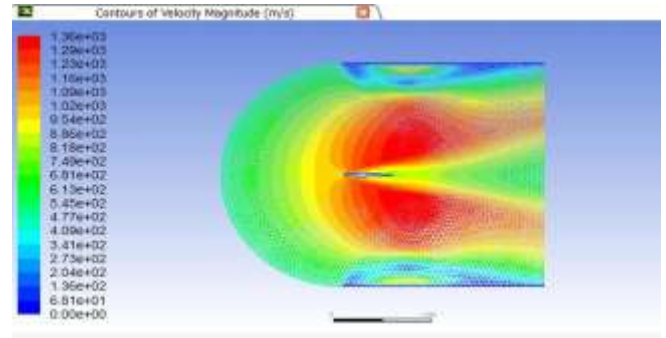


Figure: Leading Edge Radius 5 – 0 Degree AOA - Velocity Contour

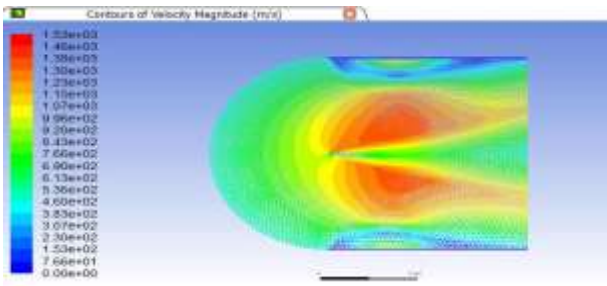


Figure: Leading Edge Radius 5 – -5 Degree AOA - Velocity Contour

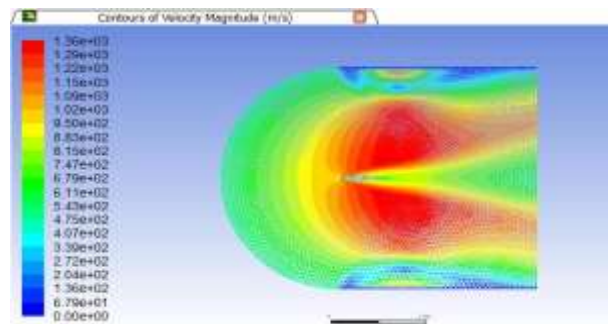
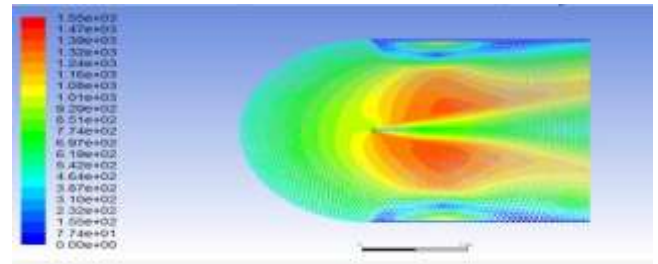


Figure: Leading Edge Radius 5.2 – 0 Degree AOA-Velocity Contour

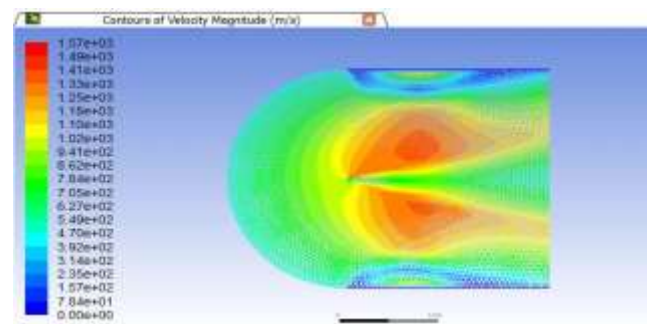


Figure: Leading Edge Radius 5.2 – 5 Degree AOA - Velocity Contour

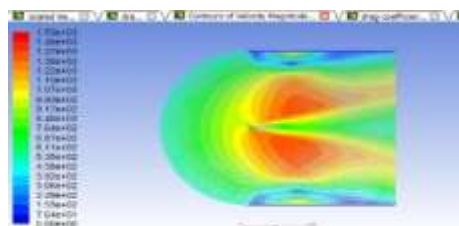


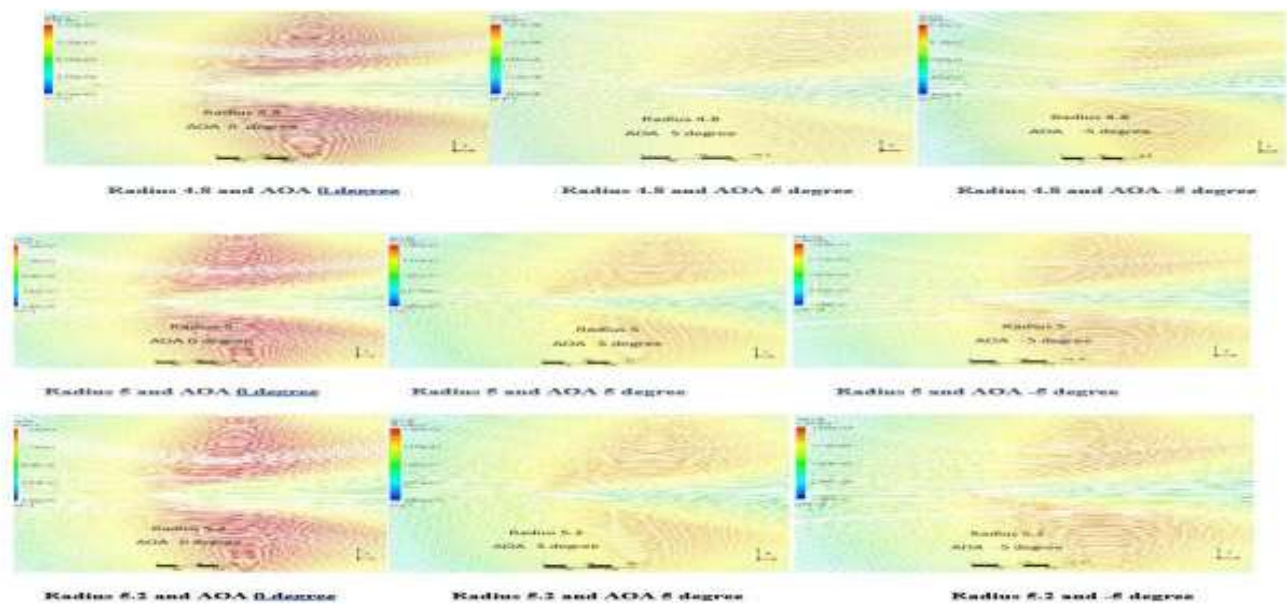
Figure: Leading Edge Radius 5.2 – -5 Degree AOA - Velocity Contour

Discussion

In contrast to the velocity distribution for negative angle of attack, we can clearly see that the velocity above the aerofoil is greater than the velocity below the aerofoil for positive angle of attack. According to Bernoulli's principle in fluid dynamics, a change in velocity induces a change in pressure. The lift force acts perpendicular to the flow direction due to the pressure difference above and below the aerofoil. As entering air strikes the aerofoil perpendicularly, a low velocity region forms at the leading edge. The aerofoil is subjected to a drag force due to friction and other causes.

1. velocity stream line

The velocity stream line obtained for angles of attack 0° , 5° and -5° around the aerofoil is shown below. The findings from these velocity stream line contours for varied angle of attacks are further explained.



Discussion

The accompanying diagram shows the velocity stream line around an aerofoil with various angles of attack. As we can see from the velocity counter, the velocity above the aerofoil is greater than the velocity below the aerofoil; similarly, the flow of velocity on the aerofoil is greater than the lower surface of the aerofoil in the velocity stream line. And there are 300 stream lines in total that we've used for this.

CONCLUSION

This paper has mainly focused on finding the lift and drag coefficients obtained for a NACA 64008 aerofoil at a supersonic speed of Mach 1.5 when the flow angle was at degree 5, 0 and -5 for the three different leading edge radius are 5, 5.2 and 4.8. Propagation of shock waves was studied. It is analysed that the flow velocity greatly influences the characteristics of the boundary layer formed.

Parameters like lift and drag forces, coefficient of lift and drag after analysing are summarized in the table 3. as the free stream velocity increases along the surfaces of the aerofoil, critical M is reached at some point after which shock

was formed along the trailing edge.

It clearly shows that for the increase in leading edge radius as considerable effect in the increase of coefficient that is for 5.2 leading edge radius for AOA positive 5 degree it clearly shows that there is an increase in lift and decrease in drag. Thus from this it is clearly observed that increase in leading edge radius as considerable effect in the increase in coefficient of lift

References

- [1] Mahasidha R. Birajdar, Sandip A. Kale, S. N. Sapali, “Numerical Analysis of New Airfoils for Small Wind Turbine Blade”, Journal of Alternate Energy Sources and Technologies, ISSN: 2230-7982, Volume 6, Issue 1, 2015. [doi.org/10.1115/IMECE2015-51681]
- [2] Mahasidha R. Birajdar, Sandip A. Kale, “Comparison of new designed small wind turbine airfoils and blade performance using different techniques”, International Journal of Applied engineering research, ISSN: 0973-4562, Volume 10, No.71, 2015.
- [3] Ronit K. Singh, M. Ra.uddinAhmeda, Mohammad AsidZullah, Young-Ho Lee, “Design of a low Reynolds number airfoil for small horizontal axis wind turbines”, Renewable Energy 42 (2012) 66-76. [doi.org/10.1016/j.renene.2011.09.014]
- [4] AgrimSareen, Chinmay A. Sapre and Michael S. Selig, “Effects of leading edge erosion on wind turbine blade performance”, Wind Energ.2014; 17:1531–1542. [doi.org/10.1002/we.1649]