RAGHU ENGINEEIRNG COLLEGE

Department of Mechanical Engineering

CERTIFICATE

This is to certify that this thesis entitled "ANALYSIS OF AN AIRFOIL USING ANSYS" is a bonafide record of the work done, in partial fulfilment of the requirement for the award of the degree of **Bachelor of Technology** in the department of Mechanical Engineering**,** Raghu Engineering College Dakamarri, Visakhapatnam.

C.BHARAT SURYA	(14981A03B0)
S.DIVAKAR	(15985A0343)
S.KIRANMAI	(14981A03D1)
J.RAMAKRISHNA	(14981A03A9)
J.NAGESWARA REDDY	(14981A03B8)
K.VENKAIAH	(14981A03A6)

Project Guide

Mr. Manish Yadav

Asst. Professor

Department of Mechanical Engineering,

Raghu Engineering College

Head of the Department

Mr. R.Vaikunta Rao

Professor and Head of Department

Department of Mechanical Engineering,

Raghu Engineering College

ACKNOWLEDGEMENT

We owe a great lot of thanks to many people who have helped us and supported us during the course of our project.

We are very grateful to our guide **Mr. Manish Yadav, Asst. Prof w**ho spent a lot of their precious time with us on our work always even though being very busy persons with the academic works of the department, for their background knowledge and patience in the completion of our project. We are very grateful to them for enlightening intellectually and spending a lot of their precious time with us even though they got a very hectic schedule. And also we extend our sincere thanks for their extreme guidance and support during the entire project. Their constant encouragement and suggestions helped us a lot. We have whole heartedly enjoyed the challenge of examining and researching a controversial issue that might make an impact on the Design and manufacturing industry instead of just performing another educational study.

We are grateful beloved HOD **R Vaikunta Rao, Professor,** for giving us access to all the facilities in the college at any odd hours of time.

We also thank all the **other faculty** and **support staff** of the Mechanical Department who have helped us in doing work with ease and perfection.

We would also thank our institution, **Raghu Engineering College,** for providing the facilities especially the CAD/CAM lab without which our project would not have been complete.

ABSTRACT

Aerodynamics has been one of the areas of design which has captured the interest of engineers over the past decade. Aerodynamic design of an aircraft includes the design of the cross section of its wing, generally known as Airfoil. The shape of an airfoil depicts the drag and lift produced which stabilises the aircraft in motion. The design of an airfoil, in recent years has been primarily initiated through a CFD analysis. The analytical results are then incorporated into the actual design.

In this report, Different airfoil shapes such as, symmetric and asymmetrical airfoils have been explored. The design data required for the airfoils have been accessed through the database of UIUC university (aerospace department) of Illinois. These shapes have been completely generated and meshed in ANSYS and CFD analysis has been carried out in ANSYS FLUENT module.

The analysis of the two dimensional subsonic flow over NACA airfoils at various angles of attack and operating at a Reynolds number of 5×E+4 has been simulated and pressure variance data has been provided in detail. Also asymmetrical airfoils shapes of commercial aircrafts such as BOEING have been analysed. The CFD analysis over selected shapes with varying input parameters such as the angle of attack(AOA) viscosity, Reynolds number, operating pressure has been done.

The simulations provided the necessary data to calculate, the drag and lift forces for different airfoil shapes. These results can be validated further by experimental testing in wind tunnel

TABLE OF CONTENTS

ABSTRACT		I
LIST OF TABLES		IV
LIST OF FIGURES		V
LIST OF GRAPHS		VI
LIST OF NOTATIONS		VII
		Page no.
CHAPTER – 1	**INTRODUCTION**	1
1.1	General	1
1.2	History	2
1.3	Aerodynamic forces	3
1.4	Theories of lift	6
1.5	Airfoil terminology	11
1.6	Airfoil Designation	12
1.7	Types of airfoils	13
1.8	Varying parameters	17
1.9	Objectives of the study	17
CHAPTER 2	**LITERATURE REVIEW**	18
2.1	General	18
2.2	Literature on Airfoil analysis	18
2.3	Concluding Remarks on Literature	20

CHAPTER 3 METHODOLOGY 21

3.1 Working with ANSYS 21

3.2 Prerequisites 21

3.3 Workbench setup 23

3.3.1 Geometry 23

3.3.2 Mesh Generation 26

3.3.3 Setup and solution 30

CHAPTER 4 FLUENT ANALYSIS OF AIRFOILS 35

4.1 Analysis of BOEING 737 airfoil 35

4.1.1 Analysis results 39

4.2 Analysis of NACA 0012 symmetric airfoil 42

4.2.2 Analysis results 45

4.3 Analysis of NACA 2412 airfoil 48

4.3.1 Analysis results 50

CHAPTER 5 RESULTS VALIDATION AND CONCLUSION 53

5.1 Analysis results 54

5.2 Validation of NACA 0012 airfoil 55

5.3 Validation of NACA 2412 airfoil 56

5.4 Validation of BOEING 737 airfoil 57

5.5 Summary and conclusion 58

5.6 Future scope and research 58

REFERNCES 59

LIST OF TABLES

Table	Title	page no
1.1	Mach number	16
4.1	Pressure gradient at varying angle of attack	37
4.2	velocity magnitude at varying AOA	38
4.3	CL and CD at different AOA	39
4.4	Lift force at different AOA – BOEING	41
4.5	Drag force at different AOA –BOEING	41
4.6	NACA 0012 pressure at different AOA	44
4.7	NACA0012 cl and cd at different AOA	45
4.8	drag and lift force at varying AOA	47
4.9	Pressure gradient of NACA 2412 varying AOA	50
4.10	velocity gradient of NACA 2412 varying AOA	50
4.11	CL and Cd at different AOA	51
4.12	lift and drag forces with varying AOA	52
5.1	stall angle of attack	54
5.2	CL AND CD at stall angle of attack	54

CHAPTER 1

INTRODUCTION

1.1 GENERAL

It is a fact of common experience that a body in motion through a fluid experience a resultant force which, in most cases is mainly a resistance to the motion. Aerodynamics is one such branch of science which deals with the movement of air, especially when associating with a solid object. More precisely, it is a basis that helps us to understand the dynamic nature of objects moving in fluid (air) i.e. aeroplanes or aircrafts.

The theory behind aerodynamics is very intriguing and captivating. One such part of theory is the generation of lift which makes an aeroplane fly. This phenomenon is greatly attributed with an important part of aeroplane, known as Airfoil.

In general terms, an airfoil (in American English) or aerofoil (in British English), is the cross section of the wing when viewed from the sides.

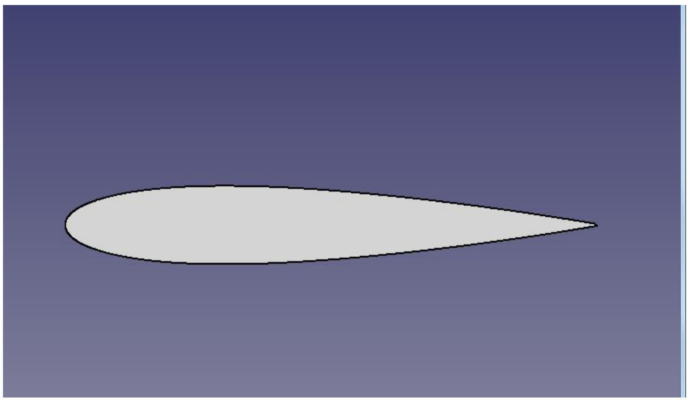


Fig 1.1 Airfoil (cross section of the wing)

An airfoil-formed body travelled through a fluid handles an aerodynamic energy. The energy possessed is distributed along distinct directions and makes an aeroplane fly. The shape of an airfoil creates a pressure and velocity gradient along its profile which is

2majorly responsible for the lift that is required for an airplane to fly. This leads to a conclusion that the design of an airfoil is a necessary step in mastering aerodynamics.
In brief, airfoil design is attributed with understanding the parameters such as flow (subsonic), pressure, velocity, geometry of the airfoil etc. hence, airfoil design has been one of the complex accomplishments in modern era.

1.2 HISTORY

Airfoil design has progressed considerably over the past century. The first airfoils were mere copies of bird's wings. The vast majority of the early exertions in aerodynamics worked towards attaining heavier-than-air flight, which was initially exhibited by Wilbur and Orville Wright in 1903. From that point forward, the utilization of aerodynamics through scientific examination, observational estimates, wind tunnel experimentation, and workstation recreations has framed the investigative premise for progressing improvements in heavier-than-air flight and various different advances. These airfoils were followed by cut-and-try shapes, some of which were tested in simple, Low-Reynolds-number wind tunnels. The National Advisory Committee for Aeronautics (NACA) systematized this approach by perturbing successful airfoil geometries to generate series of related airfoils.

By 1920, NACA has initiated the development of airfoils at the Langley Memorial Aeronautical Laboratory. The first series of airfoils, designated “M sections” This series was significant because it represented a systematic approach to airfoil development combining relations between angle of attack and pressure distribution. The empirical approach, which involved modifying the geometry of an existing airfoil, culminated in the development of the four- and five-digit-series airfoils in the mid 1930’s.

Over the course of time, pressure gradients required were mathematically transformed into airfoil shapes. This resulted in NACA2, NACA7 and most notably NACA 6 series. The NACA opted out of the airfoil business in the 1950’s, leaving the world with a large number of systematically designed and experimentally tested airfoils.

NASA then undertook the investigations and developed the airfoils suitable to sustain transonic turbulent low speed conditions. The main objective was to develop a sustainable airfoil design that would perform well in real time turbulent conditions with high lift coefficient. This resulted in NACA 4 and NACA 5-digit series'

The airfoil scene then shifted to Germany where F. X. Wortmann and Richard Eppler were engaged in laminar-flow airfoil design. Eastman Jacobs of NACA recognized the need for a theoretical method that would determine the airfoil shape that would produce a specified pressure distribution that would exhibit the desired boundary-layer characteristics airfoil shape.

Thus, the basic idea behind modern airfoil design was conceived: the desired boundary layer characteristics result from the pressure distribution which results from the airfoil shapes.

1.3 AERODYNAMIC FORCES

When an aeroplane is in motion, four forces generally accompany it.

They are:

LIFT

WEIGHT

DRAG

THRUST

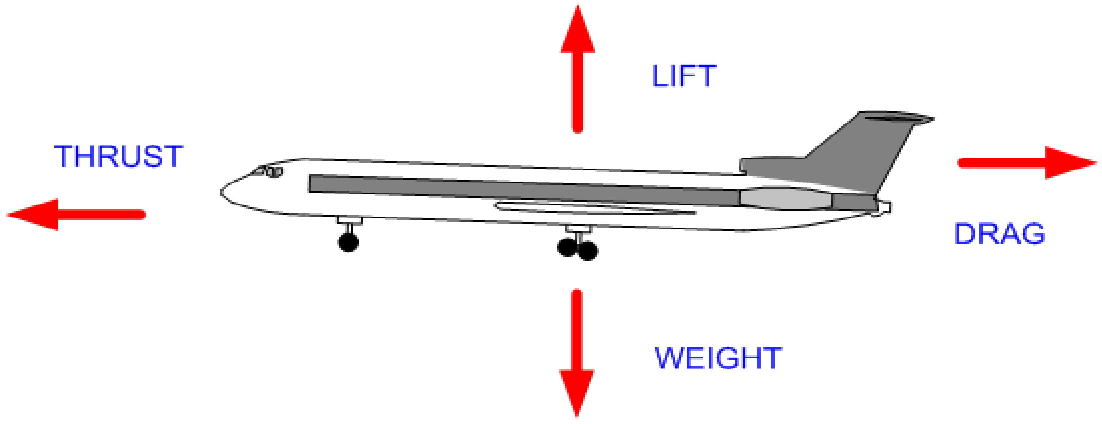


FIG 1.2 AERODYNAMIC FORCES ON AN AIRPLANE

1. LIFT

As mentioned earlier, airfoil is the cross section of the wing .Its job is to provide necessary lift opposing the weight during take-off and in flight.

In the preceding sections, it has been established that aerodynamic energy is developed by the body in motion. The segment of energy perpendicular to the course of movement is called **lift**.

For a stable flight, the lift generated must be sufficient to overcome the weight of the aeroplane. The amount of lift needed by a plane depends on the purpose for which it is to be used. Heavier planes require more lift while lighter planes require less lift than the heavier ones. Thus, depending upon the use of aeroplane, airfoil section is determined. Lift force also determines the vertical acceleration of the plane, which in turns depends on the horizontal velocity of the plane. Thus, determining the coefficient of lift one can calculate the lift force and knowing the lift force and required vertical acceleration one can determine the required horizontal velocity.

2. DRAG

It is a well-known fact frictional forces oppose a solid body in motion on a rough surface. The drag force is analogous to the above phenomena. Hence the drag offered by air is nothing but aerodynamic friction in simple terms. Here the medium is fluid(air), and object in motion is aeroplane.

More precisely, the segment of energy parallel to the bearing of movement is called **drag.** the drag is the opposing force that counteracts the thrust of the engine.

There are many factors which influence the amount of aerodynamic drag which a body generates. Drag depends on the shape, size, and inclination, of the object, and on flow conditions of the air passing the object. For a three dimensional wing, there is an additional component of drag, called **induced drag**,

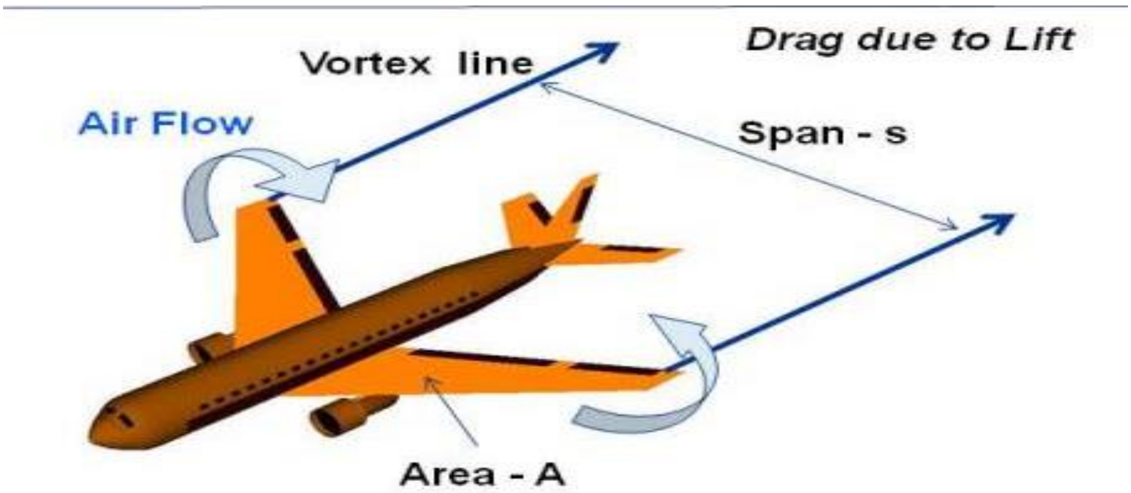


FIG 1.3 INDUCED DRAG ON AN AIRPLANE

For a lifting wing, the air pressure on the top of the wing is lower than the pressure below the wing. Near the tips of the wing, the air is free to move from the region of high pressure into the region of low pressure the blue arrows show the flow direction of vortices induced. As the aircraft moves to the lower left, a pair of counter-rotating vortices are formed at the wing tips. The line of the centre of the vortices are shown as blue vortex lines leading from the wing tips. The wing tip vortices produce a swirling flow of air behind the wing which is very strong near the wing tips and decreases toward the wing root. The effective angle of attack of the wing is decreased by the induced flow of the vortices and varies from wing tip to wing root. The induced flow produces an additional, downstream-facing, component of aerodynamic force of the wing. This additional force is called **induced drag** because it faces downstream and has been "induced" by the action of the tip vortices. It is also called "drag due to lift" because it only occurs on finite, lifting wings and the magnitude of the drag depends on the lift of the wing.

From the above statements be inferred that, drag and lift are two important aerodynamic forces that decide the flight of an object.

During cruise, **Lift = (WEIGHT=mg)**

Drag< thrust.

Since drag is an unwanted force, it has to be minimised by increasing the lift.

1.4 THEORIES OF LIFT

Mainly two theories are more prevalent that explain the lift satisfactorily, although there are certain limitations, for design purpose basic understanding of these theories is sufficient.

For a body immersed in a moving fluid, the fluid remains in contact with the surface of the body. If the body is shaped, moved, or inclined in such a way as to produce a net deflection or turning of the flow, the local velocity is changed in magnitude, direction, or both. Changing the velocity creates a net force on the body. It is very important to note that the turning of the fluid occurs because the molecules of the fluid stay in contact with the solid body since the molecules are free to move.

Any part of the solid body can deflect a flow. Parts facing the oncoming flow are said to be **windward**, and parts facing away from the flow are said to be **leeward**. Both windward and leeward parts deflect a flow. Ignoring the leeward deflection leads to a popular incorrect theory of lift.

1 BERNOULLI'S THEORY

The lift on an airfoil is fundamentally the consequence of its approach and shape. Also called longer path or equal transit theory,it predicts that air velocity at the upper curved surface is greater than lower surface so as to have equal transition time in meeting at the trailing edge.

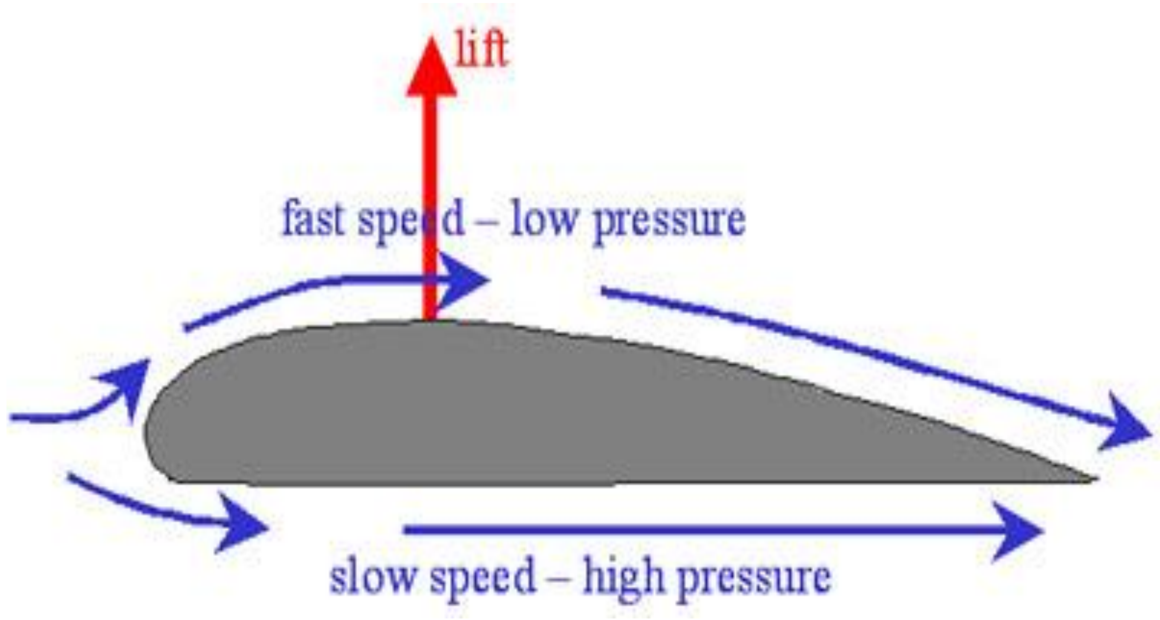


FIG 1.4 BERNOULLI'S LIFT

As the velocity gradient is observed, pressure gradient comes into the picture resulting in lift upwards. The major assumption here is air flows in streamlines and gets deflected by leading edge of airfoil.

So mathematically, bernoullis explantion can be put forward in the following way:

$$p + \frac{1}{2}\rho V^2 + \rho g h = constant$$

p- pressure, g-gravity, h-elevation , V-velocity, ρ-density.

Drawback:

The drawback of this theory is its explanation of velocity gradient. Experiments show that pressure difference certainly creates lift but the air deflected doesn't come into contact at the trailing edge in equal time. The air above is more faster than anticipated.

2 SKIPPING STONE THEORY

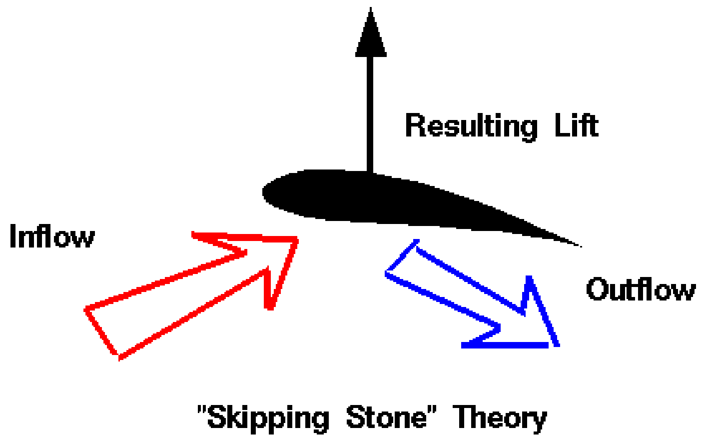


FIG 1.5 NEWTONIAN MODEL OF LIFT

The theory is based on the idea that lift is the reaction force to air molecules striking the bottom of the airfoil as it moves through the air. Because this is similar to the way in which a flat rock thrown at a shallow angle skips across a body of water, it is called the "Skipping Stone" theory of lift. It is sometimes called a Newtonian theory of lift.

Drawbacks: This theory is concerned with only the interaction of the lower surface of the moving object and the air. It assumes that all of the flow turning (and therefore all the lift) is produced by the lower surface. But the upper surface also turns the flow. This theory does not predict or explain this effect.

- Because this theory neglects the action - reaction of molecules striking the upper surface, it does not predict the negative lift present when the angle of attack is negative. On the top of the airfoil, no vacuum exists. Molecules are still in constant random motion on the upper surface (as well as the lower surface), and these molecules strike the surface and impart momentum to the airfoil as well.

3 ACCEPTED EXPLANATION:

The real details of how an object generates lift are very complex and do not lend themselves to simplification. For a gas, we have to simultaneously conserve the mass, momentum, and energy in the flow. Newton's laws of motion are statements concerning the conservation of momentum. Bernoulli's equation is derived by considering conservation of energy. So both of these equations are satisfied in the generation of lift; both are correct.

The conservation of mass introduces a lot of complexity into the analysis and understanding of aerodynamic problems. The simultaneous conservation of mass, momentum, and energy of a fluid (while neglecting the effects of air viscosity) are called the Euler Equations . If we include the effects of viscosity, we have the Navier-Stokes Equations which are named after two independent researchers in France and in England.

At the point when arranged at a suitable edge, the airfoil diverts the approaching air, bringing about an energy on the airfoil in the heading inverse to the diversion. This power is known as aerodynamic drive and could be determined into two parts: Lift and drag. This "turning" of the air in the region of the airfoil makes bended streamlines which brings about more level weight on one side and higher weight on the other. This weight contrast is joined by a speed distinction, through Bernoulli's standard, so the ensuing stream field about the airfoil has a higher normal speed on the upper surface than on the more level surface. The lift power might be connected specifically to the normal top/base speed contrast without registering the weight by utilizing the idea of flow. Lift can be generated by a wide variety of objects, including airplane wings, rotating cylinders, spinning balls, and flat plates. Lift is the force that holds an aircraft in the air. Lift can be generated by any part of the airplane, but most of the lift on a normal airliner is generated by the wings. How is lift generated?

4 FORCE OF LIFT

Lift is a force. From Newton's second law of motion, a force **F** is produced when a mass **m** is accelerated **a**:

Force = Mass x Acceleration

$$F = m * a$$

An **acceleration** is a change in velocity **V** with a change in time **t**.

$$F = m * (V1 - V0) / (t1 - t0)$$

We have written this relationship as a difference equation, but it is recognized that the relation is actually a differential from calculus.

$$F = m * dV/dt$$

The important fact is that a force causes a change in velocity; and, likewise, a change in velocity generates a force. The equation works both ways. A velocity has both a **magnitude** called the speed and a direction associated with it. Scientists and mathematicians call this a vector quantity. So, to change either the speed or the direction of a flow, you must impose a force. And if either the speed or the direction of a flow is changed, a force is generated.

1.5 AIRFOIL TERMINOLOGY

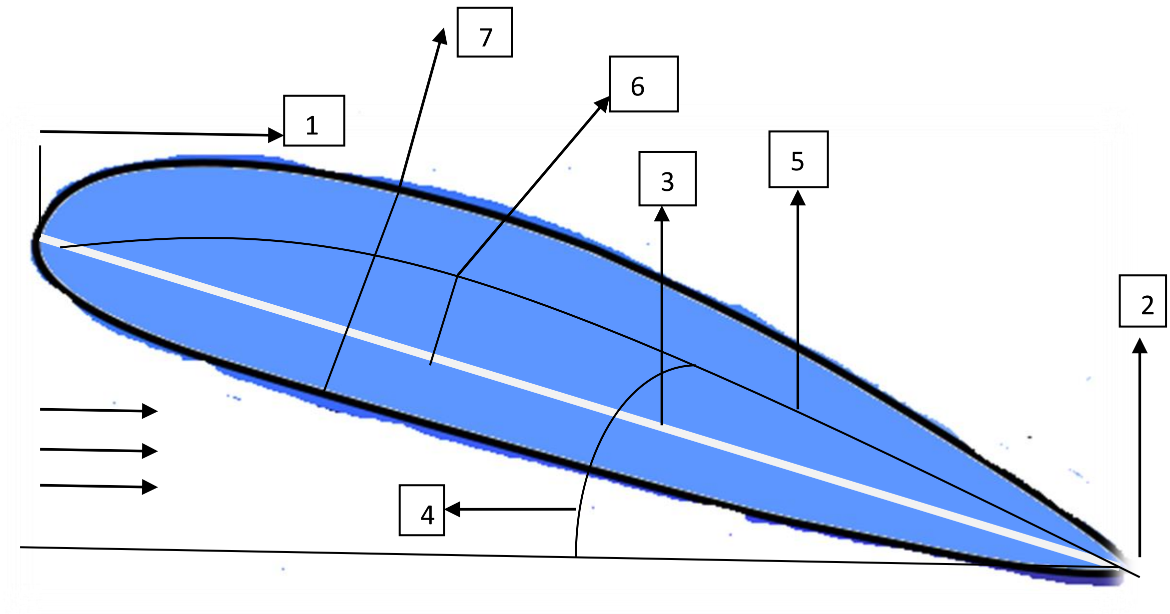


FIG 1.6 AIRFOIL TERMINOLOGY

Some terms related to Airfoil are:

1 Leading edge: - It is the edge of the airfoil facing the direction of motion of plane. It is generally roundish in shape and deflects the air in such a way that the velocity of air on upper surface of the airfoil is more than velocity on the lower surface.

2 Trailing edge :- It is the edge of the airfoil which is pointed in nature. It is located at the back side of the airfoil.

3 Chord line :- It is a straight line joining the leading edge to the trailing edge. It bisects the airfoil into two parts for an symmetric airfoil but may not do so for an asymmetric airfoil. It defines another important parameter Angle of attack.

4 Angle of attack :- It is the angle which the chord line makes with the direction of motion of plane. It is an important parameter which affects the coefficient of lift and drag.

5 Camber line :- It is a line joining leading edge and trailing edge and dividing the airfoil into two symmetrical parts. It may or may not be a straight line.

6 Camber: the maximum distance between the chord and the camber line .expressed in %age of chord. For eg (max camber at 2% of chord)

7 Thickness: The maximum distance between the upper and lower surfaces expressed in terms of %age of chord

Stall angle of attack: - It is the angle of attack at which the lift coefficient is maximum and after which the lift coefficient starts to decrease.

Lift coefficient: - It is a dimensionless coefficient that relates the lifting force on the body to its velocity, surface area and the density of the fluid in which it is lifting.

Drag coefficient: - It is a dimensionless coefficient that relates the dragging force on the body to its velocity, surface area and the density of the fluid in which it is moving. Stall

Because lift and drag are both aerodynamic forces, the ratio of lift to drag is an indication of the aerodynamic efficiency of the airplane. Aerodynamicists call the lift to drag ratio the **L/D ratio**, pronounced.

An airplane has a high L/D ratio if it produces a large amount of lift or a small amount of drag. Under cruise conditions lift is equal to weight. A high lift aircraft can carry a large payload. Under cruise conditions thrust is equal to drag. A low drag aircraft requires low thrust.

1.6 AIRFOIL DESIGNATION

Depending on the geometry of airfoil, the drag and lift induced in an aircraft vary. This report mainly focuses on NACA 4-digit series airfoils.

The NACA airfoil series is designated by 4 digits e.g. NACA 2414, which designate the camber, position of the maximum camber and thickness. If an airfoil number is

NACA MPXX

e.g.

NACA 2414

- M is the maximum camber divided by 100. In the example M=2 so the camber is 0.02 or 2% of the chord

- P is the position of the maximum camber divided by 10. In the example P=4 so the maximum camber is at 0.4 or 40% of the chord.
- XX is the thickness divided by 100. In the example XX=14 so the thickness is 0.14 or 14% of the chord.

1.7 TYPES OF AIRFOILS:

Airfoils can be basically categorized as symmetric**,** asymmetric, cambered.

In a **symmetric** airfoils, the chord line divides the profile equally in areas.

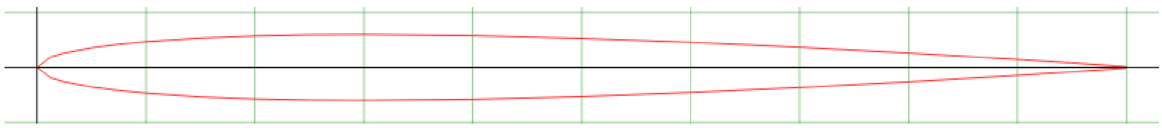


Fig 1.7 NACA 0006 SYMMETRIC AIRFOIL

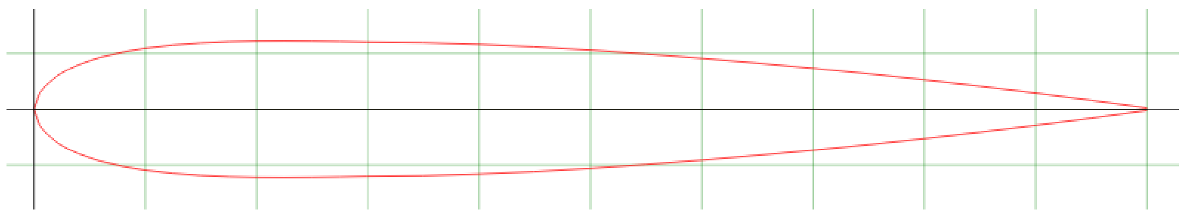


FIG 1.8 NACA 0012 SYMMETRIC AIRFOIL

In this report we have analysed the above mentioned airfoils in the segment of symmetric airfoil.

Symmetric airfoils show the same pressure distribution over both its profiles. Therefore, varying the angle of attack results in pressure variation. However, because of its low coefficient of lift with varying angle of attack, these have become obsolete, (only being used for domestic propeller planes such as Lockheed.) and asymmetric airfoils have replaced them.

In an **asymmetric** airfoil the camber lines divides the profile into equal areas. hence, in an asymmetric airfoil, there is a positive value of camber.

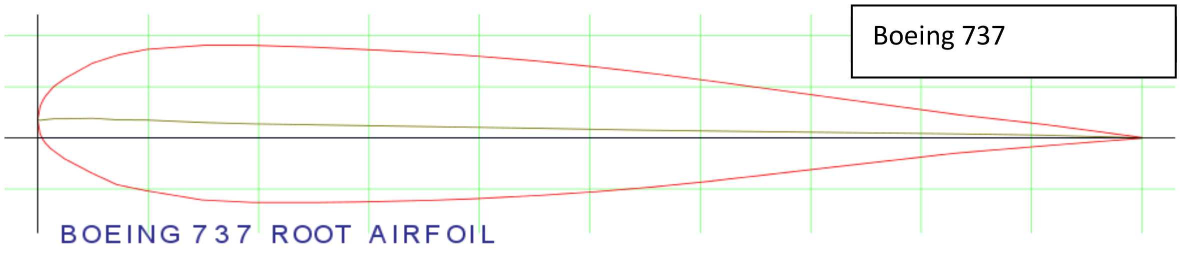


FIG 1.9 BOEING 737 AIRFOIL

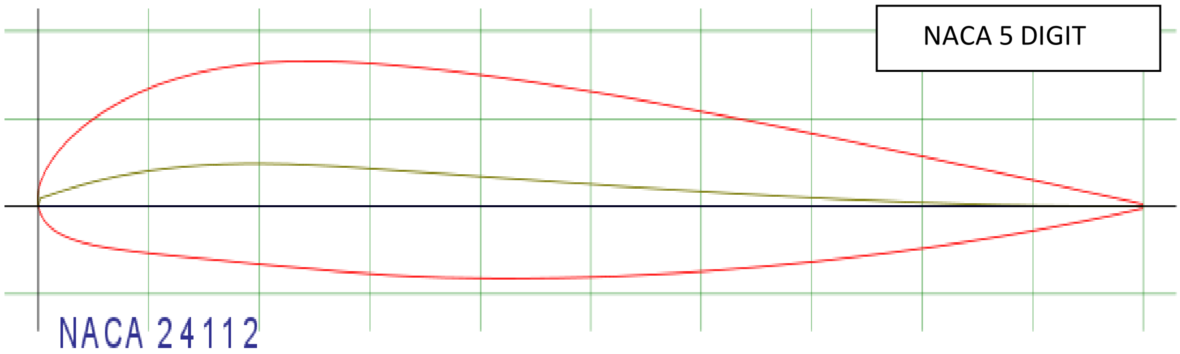


FIG 1.10 NACA 24112 AIRFOIL

In this report, commercially used airfoil design such as Boeing 737 has been analysed as well.

The terminology of those airfoils is mentioned in methodology section.

However, these data are only approximations of actual design since the data regarding actual design is highly confidential and often a trade secret of manufacturer. Using soft wares such as ANSYS , the profiles are modelled and meshed. Boundary conditions are imposed on the setup and input parameters are varied. Accordingly, real time simulation with varying angle of attack is done. The results are tabulated inn further sections

1.8 VARYING PARAMETERS:

Generally aircrafts manoeuvre in subsonic flows having mach number around 0.6

Mach number is defined as velocity of the medium or body to velocity osf sound in air.

$$M = v / c$$

where

M = Mach number

v = fluid flow speed (m/s, ft/s)

c = speed of sound (m/s, ft/s)

compressibility effects:

whenever the properties such as fluid density change, the flow i s said to be compressible. For the same reason, we follow the continuum hypothesis which suggests that molecules of air remain incompressible to greater extent. Therefore while dealing with simple fluid mechanics problems we consider the fluid to be incompressible in most of the case. However, while dealing with gas dynamics, assuming it as a standard case of reality, gases mostly exhibit compressibility. The extent of compressibility is given by compressibility factor.

However, flows are usually treated as being incompressible when the Mach number (the ratio of the speed of the flow to the speed of sound) is less than 0. The study of compressible flow is relevant to high-speed aircraft, jet engines, rocket motors, high-speed entry into a planetary atmosphere, gas pipelines, commercial applications such as abrasive blasting, and many other fields.

The Mach number is primarily used to determine the approximation with which a flow can be treated as an incompressible flow. The medium can be a gas or a liquid. The boundary can be travelling in the medium, or it can be stationary while the medium flows along it, or they can both be moving, with different velocities: what matters is their relative velocity with respect to each other. The boundary can be the boundary of an object immersed in the medium, or of a channel such as a nozzle, diffusers or wind tunnels channel in the medium. As the Mach number is defined as the ratio of two speeds, it is a dimensionless number.
If $M < 0.2–0.3$ and the flow is quasi-steady and isothermal, compressibility effects will be small and simplified incompressible flow equations can be used.

Mach number	**flow**
<0.8	Subsonic (our analysis)
0.8-1.3	transonic
1.3-5.0	supersonic
>5.0	Hypersonic

Table 1.1 Mach number

Factors affecting the performance of an airfoil:

- Viscosity
- Chord length
- Reynolds number
- Velocity of aircraft

- Viscosity determines the drag that is exhibited. Turbulent flow produces vortices which in turn produces induced drag.

- Reynolds number is a dimensionless number which decides the flow is laminar or turbulent. Our analysis is carried out at Reynolds number of 5xe-6.
- Sine we deal with Mach around 0.8, flow can be considered to be viscous and incompressible.
- NACA airfoils generally are designed for turbulent flow and therefore with varying Reynolds flow, the analysis is carried out.

1.9 OBJECTIVES OF THE STUDY

- Modelling of Symmetric and Asymmetric Aerofoils in ANSYS.
- Modelling through inputs of coordinate database and surface generation of NACA2412, NACA0012
- Modelling of air foils of BOEING737,
- Examine the velocity and surface pressure distribution of various foils.
- Determine the drag and lift coefficient of the various Airfoils and study their variation with angle of attack.
- Plot the contours of pressure, velocity at various angle of attack
- Validate the results using XFoil prescribed approximations.

CHAPTER 2

LITERATURE REVIEW

2.1 GENERAL

The design of an airfoil has been one of the most demanding areas of aerodynamic design. As mentioned, it majorly influences the lift generated to support the weight of an aircraft, thereby affecting its performance. Simulation software such as CATIA, ANSYS have been on a rise, owing to its ease of simulation in real time environment whose results very much converge with the experimental data. These have been proved as a useful tool in determining the drag, lift, Pressure and velocity variations associated with an airfoil and validating the outcome with experimental and analytical methods.

An attempt has been made to review the simulations carried out in these softwares . Based on the literature reviewed, the specific scope of work is defined.

2.2 LITERATURE ON AIRFOIL ANALYSIS

The data regarding the shape of an airfoil is highly confidential and is not generally provided by the manufactures or firms concerned. However , the department of aeronautic design (UIUC),Illinois; maintains a database which consists of the approximate design of numerous airfoils, ranging from domestic to commercial and defence.

These shapes are coded through a 4 digit, 5-digit series with a prefix as NACA (national advisory committee of aeronautics). The contours of pressure and velocity along with plots of stall angle, coefficient of drag and lift are secured in these databases.

T .Gultop, (2005) studied the impact of perspective degree on Airfoil performance. The reason for this study was to focus the ripple conditions not to be kept up throughout wind tunnel tests. These studies indicate that aero elastic insecurities for the changing arrangements acknowledged showed up at Mach number 0.55, which was higher than the wind tunnel Mach number point of confinement velocity of 0.3.

Sanjay Goel,(2008)[3] devised a method of optimization of Turbine Airfoil using Quansi – 3D analysis codes. He solved the complexity of 3D modelling by modelling multiple 2D airfoil sections and joining their figure in radial direction using second and first order polynomials that leads to no roughness in the radial direction.

Mr. Arvind (2010) researched on NACA 4412 airfoil and analysed its profile for consideration of an airplane wing. The NACA 4412 airfoil was created using CATIA V5 and analysis was carried out using commercial code ANSYS 13.0 FLUENT at an speed of 340.29 m/sec for angles of attack of 0˚, 6, 12 and 16˚. k-ε turbulence model was assumed for Airflow. Fluctuations of static pressure and dynamic pressure are plotted in form of filled contour.

Mr. Mayurkymar kevadiya (2013) studied the NACA 4412 airfoil profile and recognized its importance for investigation of wind turbine edge. Geometry of the airfoil is made utilizing GAMBIT 2.4.6. Also CFD investigation is done utilizing FLUENT 6.3.26 at different approaches from 0˚ to 12˚.

Rana et al. Studied the flutter characteristics of an airfoil in a 2-D subsonic flow by using RANS based CFD solver with a structural code in time domain.

Kevadiya concentrated on the NACA 4412 aerofoil profile and remembered its significance for examination of wind turbine edge. Geometry of the aerofoil is made using GAMBIT 2.4.6. Additionally, CFD examination is done using FLUENT 6.3.26 at distinctive methodologies from 0˚ to 12˚.

Guilmineau et al. talked about the processing of the time-mean, turbulent, two-dimensional incompressible thick stream past an airfoil at settled rate. Another physically reliable technique is exhibited for the reproduction of speed fluxes which emerge from discrete mathematical statements for the mass and energy equalization. This conclusion strategy for fluxes makes conceivable the utilization of a cell-focused network in which speed and pressure questions have the same location, while going around the event of spurious pressure modes.

Kunz and Kroo advanced aerofoils at ultra-low Reynolds numbers. These examinations are done to comprehend the aerodynamic issues identified with the low speed and miniaturized scale air vehicle outline and execution. The enhancement strategy utilized is in view of concurrent pseudo time venturing in which stationary states are acquired by fathoming the preconditioned pseudo stationary arrangement of comparisons.

N. Ahmed et al. contemplated the numerical reproduction of stream past aerofoils is vital in the flight optimized outline of air ship wings and turbo-hardware parts. These lifting gadgets regularly achieve ideal execution at the state of onset of partition. Hence, division phenomena must be incorporated if the examination is gone for pragmatic applications. Thus, in the present study, numerical recreation of relentless stream in a straight course of NACA 0012 aerofoils is expert with control volume approach

Mittal et al. performed the computational examination for two-dimensional stream past stationary NACA 0012 aerofoil is completed with dynamically expanding and diminishing approaches. The incompressible, Reynolds averaged Navier–Stokes mathematical statements in conjunction with the Baldwin–Lomax model, for turbulence conclusion, are explained utilizing settled limited element formulations. Several experimental studies were also conducted to understand the dynamic behaviour and flow characteristics of aerofoil sections.

Genc et al conducted experiments on NACA 2415 aerofoil by varying angle of attack from -120 to 200 at low Reynolds number flight regime (0.5x105 to 3x105). Using pitot static tube, scan valve and pressure transducer, the pressure distribution over aerofoil was measured. Lift, drag and pitching moment were obtained by 3-component load cell system. Hot wire anemometer and oil flow visualization was used to photograph surface flow patterns.

2.3 CONCLUDING REMARKS ON LITERATURE

A lot of work has been done of the performance of an airfoil under varying conditions. going through the works mentioned, it can be inferred that an airfoil performance is governed by factors such as viscosity, turbulent or laminar, vortices, curvature of the camber, geometry of the airfoil, chord length and so on. Our attempt is to study the pressure and velocity distributions under these conditions using ANSYS and plotting the contours of the same against varying angles of attack.

CHAPTER 3

METHODOLOGY

Carrying out an analysis in ANSYS involves four basic levels. Each level has to be made free from errors in order to proceed with the next level.

These four levels are:

1. Geometry (design modeller)
2. Mesh
3. Setup
4. Solution
5. Results (user defined)

3.1 WORKING WITH ANSYS:

- Advantages of ANSYS DM/AM:
  - Robust collection of meshing algorithms (AM)
  - Easily integrated with CAD packages for complex geometries (DM)
  - Geometry can be parameterized (DM/AM)
  - Variety of approximation techniques that converge at different rates

- Disadvantage of ANSYS DM/AM:
  - Blocking of geometry can be difficult (DM)
  - Difficult to analyse the options available at the first go.
  - Less control of mesh specifics, structured meshing more difficult (AM)

3.2 Pre-requisites:

Three different airfoil sections were identified. i.e.; NACA 0012, NACA 2412, BOEING 737. As a part of our objective, it has been mentioned that analysis has been carried out on these airfoils.

In order to carry out the analysis it is required to study the

1 geometry of these airfoils,

2 XFoil predictions at various Mach number,

3 Xfoil predictions at different turbulent ratios, (n critical)

4 Angle of attack

5 Chord lengths

Different eBooks have been referred to have an overview of what analysis in ANSYS is like. The basics of meshing are learnt and improvisations were done in order to obtain fine mesh that converges to already available results.

The study of airfoils was carried out specifically on NACA 2412, NACA 0012, BOEING 737.

1. **NACA 2412**

- The camber is 0.02 or 2% of the chord
- The maximum camber is at 0.4 or 40% of the chord.
- The thickness is 0.12 or 12% of the chord.

2. **NACA0012**

- The camber is at 0% of the chord(symmetric)
- The maximum camber is at 0% of the chord
- The thickness is .12 or 12% of the chord.

- ➢ The simulations of airfoil are carried out at chord length of 1m (default units set as meters)
- ➢ Xfoil predictions are obtained from UIUCdatabase/airfoiltools.com and are used for validation.
- ➢ Turbulent ratio of 10 in the ANSYS setup module represents turbulent flow to be approximating turbulence lower the ratio –higher the turbulence and vortex and more hectic is the processing time and analysis.
- ➢ As angle of attack can be varied ranging from 5 to 15 degrees (considered optimum), the analysis has been carried out at 0^0 and choosing other angle of attack arbitrarily.

3.3 WORKBENNCH SETUP

The first and foremost step in modelling and analysis by ANSYS is setting up the required interface. In our case, the required setup is **workbench**.

A workbench consists of different modules. For analysis over an airfoil we use the **FLUENT/CFX** module.

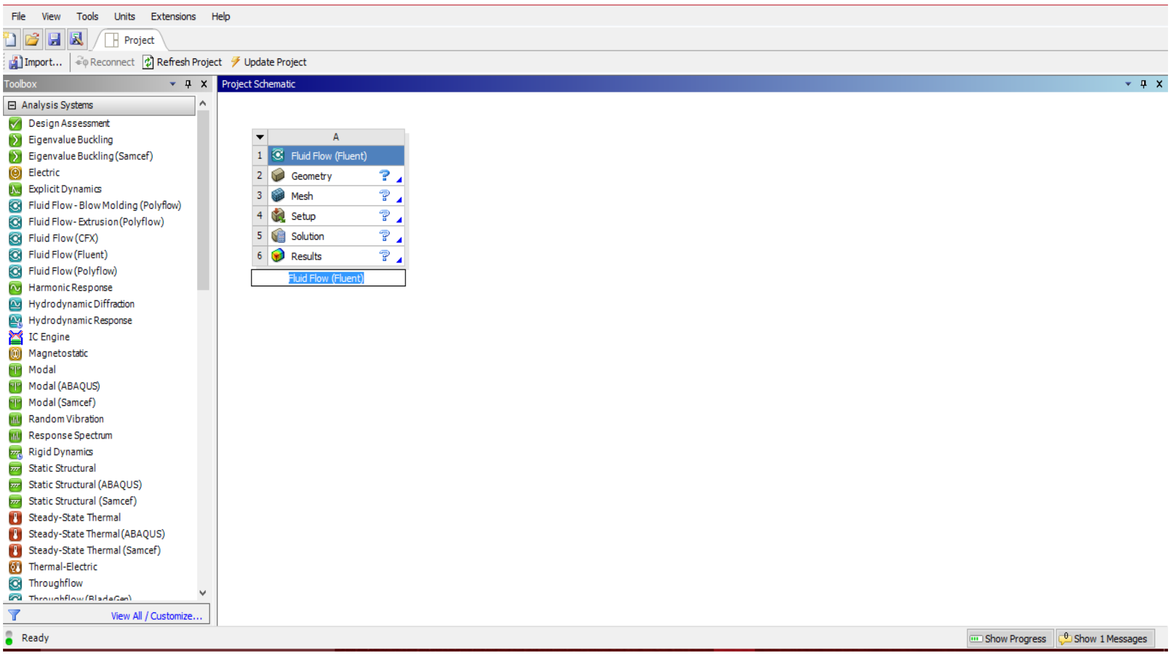


fig3.1 ANSYS WORKBENCH

A typical FLUENT module of ANSYS 16.0 is shown above. As mentioned at the start of the chapter, the four level dialogue box appears. Each stage is processed step by step.

3.3.1 GEOMETRY

In ANSYS the geometry is generated using design modeller(DM).

The sketch of airfoil profile can be generated using the coordinate points provided by the UIUC database. For reference the below explanations are made considering NACA0012 airfoil.

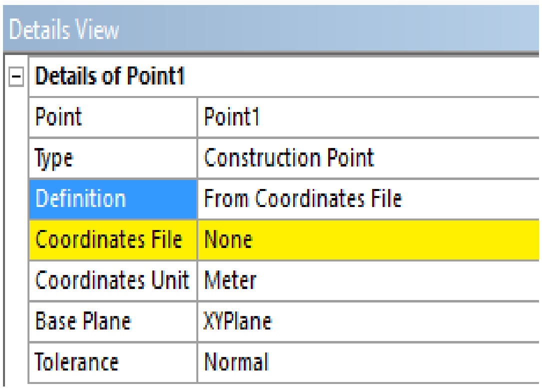
The points are imported in **selig** file format which is as follows.

Series group x co-ord y co-ord z co-ord

E.g: NACA 0012 co-ordinates

1	1	1.000000	0.001260	0
1	2	0.950000	0.008070	0
1	3	0.900000	0.014480	0

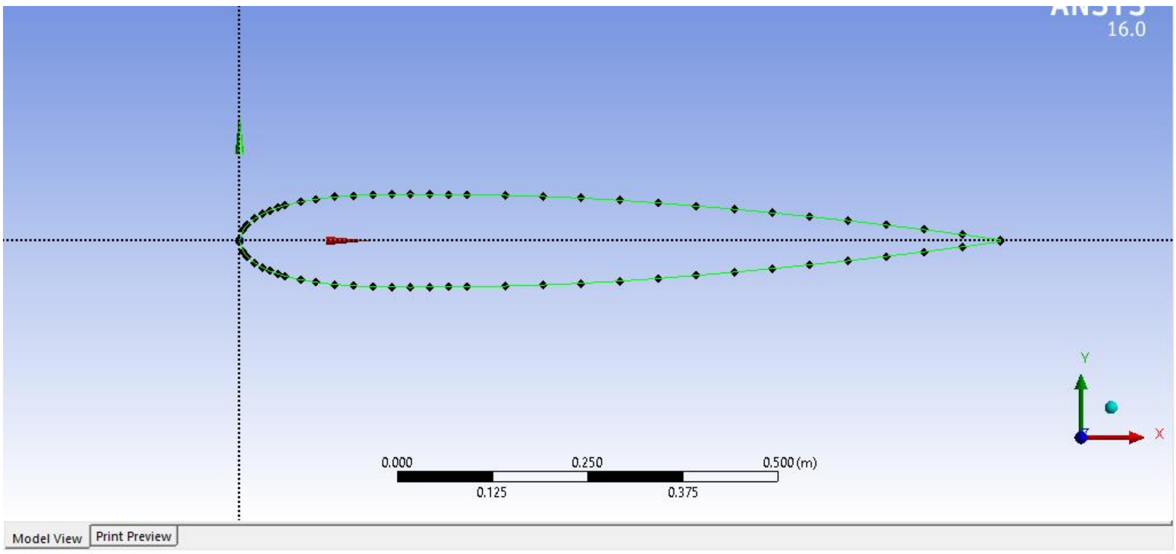
and so on…



Details View

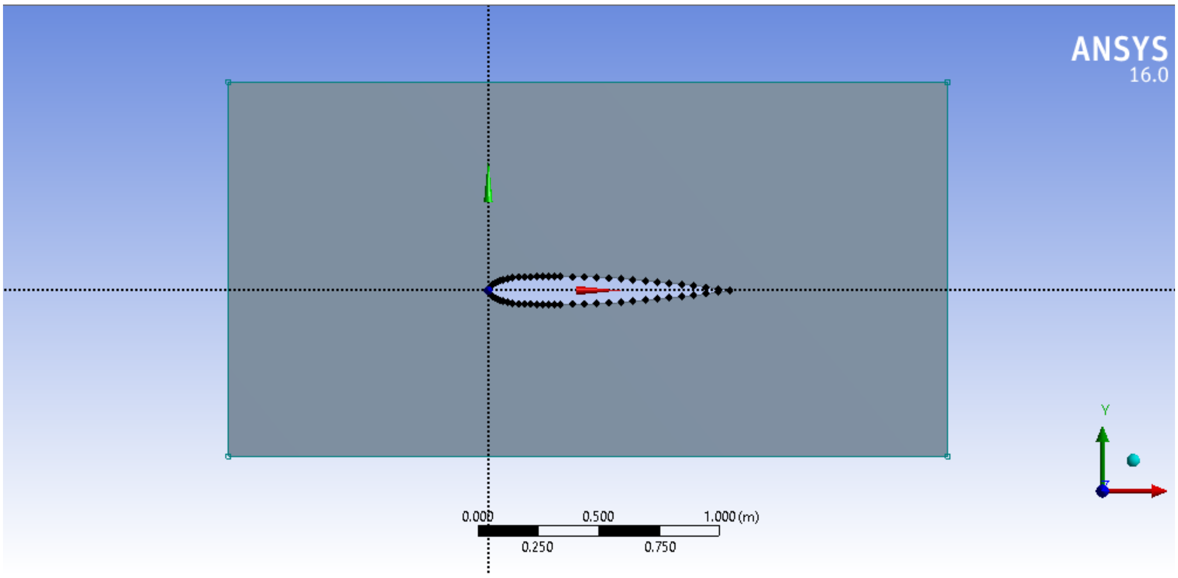
Details of Point1	
Point	Point1
Type	Construction Point
Definition	From Coordinates File
Coordinates File	None
Coordinates Unit	Meter
Base Plane	XYPlane
Tolerance	Normal

These points are then joined to produce the sketch and then surface is produced imposing Boolean operators.



AN EXAMPLE OF NACA 0012 SKETCH GENERATION

Here we see, the X axis divides the profile into two equal areas. Hence this is a symmetrical airfoil and chord lies on x axis.



WIND TUNNEL – AIRFOIL GEOMETRICAL SETUP

A wind tunnel is generally used in experimental techniques in real time simulations. Air from one end at defined properties enters the wind tunnel and flows past the airfoil to the other end. The required results are then computed.

Since we are using virtual simulations, a c-shaped wind tunnel or rectangular wind tunnel is used for the same purpose. Similar sort of boundary conditions are imposed after meshing the setup.

Angle of attack

By definition, the angle at which the airfoil is to the incoming airflow is angle of attack Using the rotate tool in modelling, the airfoil can be set at an angle to the specified axis in the global coordinate system

The similar meshing and setup procedure can be employed for varying angle of attack. The only difference lies with the orientation of the geometry with respect to the flow field.

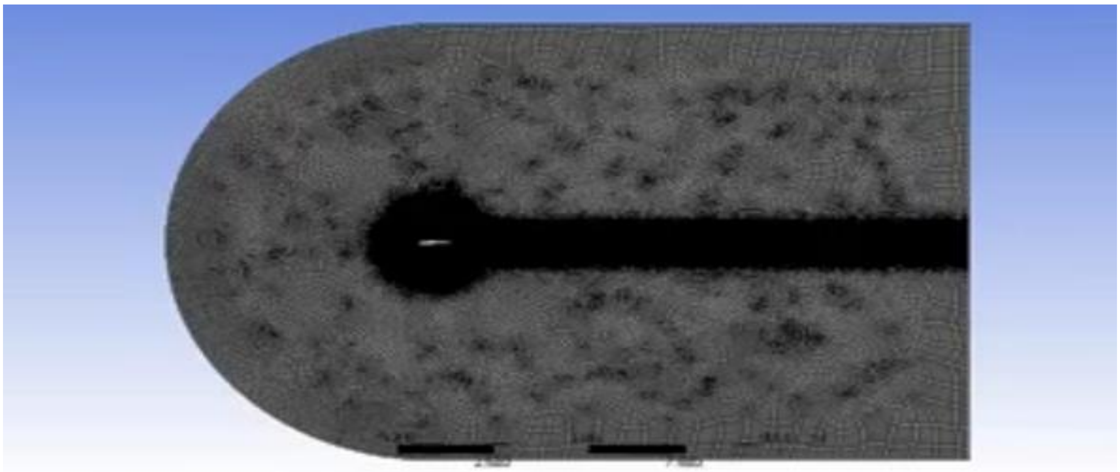
3.3.2 MESH GENERATION

Meshing is the most important step of a simulation in ANSYS. As a matter of fact, meshing is 75% of the simulation. The better the mesh we create depending upon the geometry, the better the solution converges.

- Meshing is nothing but discretization of continuous body into number of elements.
- The nodes and elements formed must be as small as possible so that loading and boundary conditions can be more accurately imposed.
- The solution converges more accurately but the processing time and CPU usage increases.
- The type of analysis being carried out demands a specific type of meshing
- Mesh can be structured, unstructured
- It can be made into hexagonal, quadrilateral, triangular elements depending on the requirement

For the mesh generation of airfoil, unstructured meshing has been employed, since it is a 2dimensional geometry.

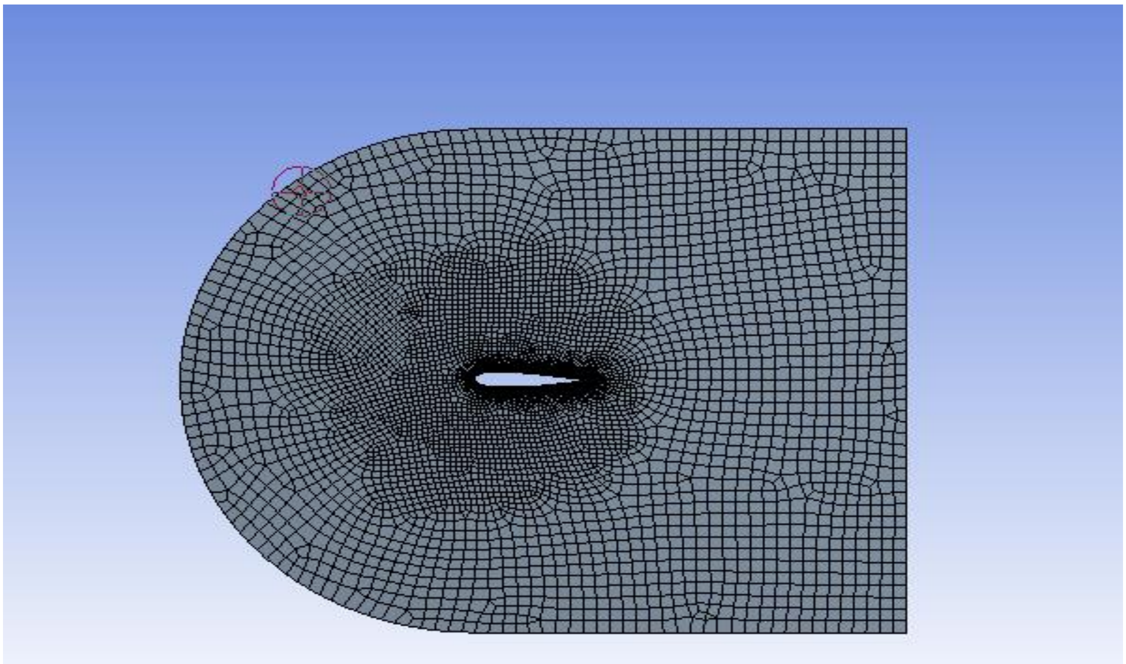
However, near the boundary of airfoil, different meshing tools have been employed to improve the meshing so as to account for the boundary layer effects and viscous drag etc.



A C MESH OVER NACA 0012

From the mentioned figure, it is observed that, the meshing becomes more complex anfd finer near the boundary of airfoil.

The number of elements near the boundary have been increased.



A CMESH OF BOEING 737

The following mesh tools have been employed

1 body sizing:

2 Edge sizing

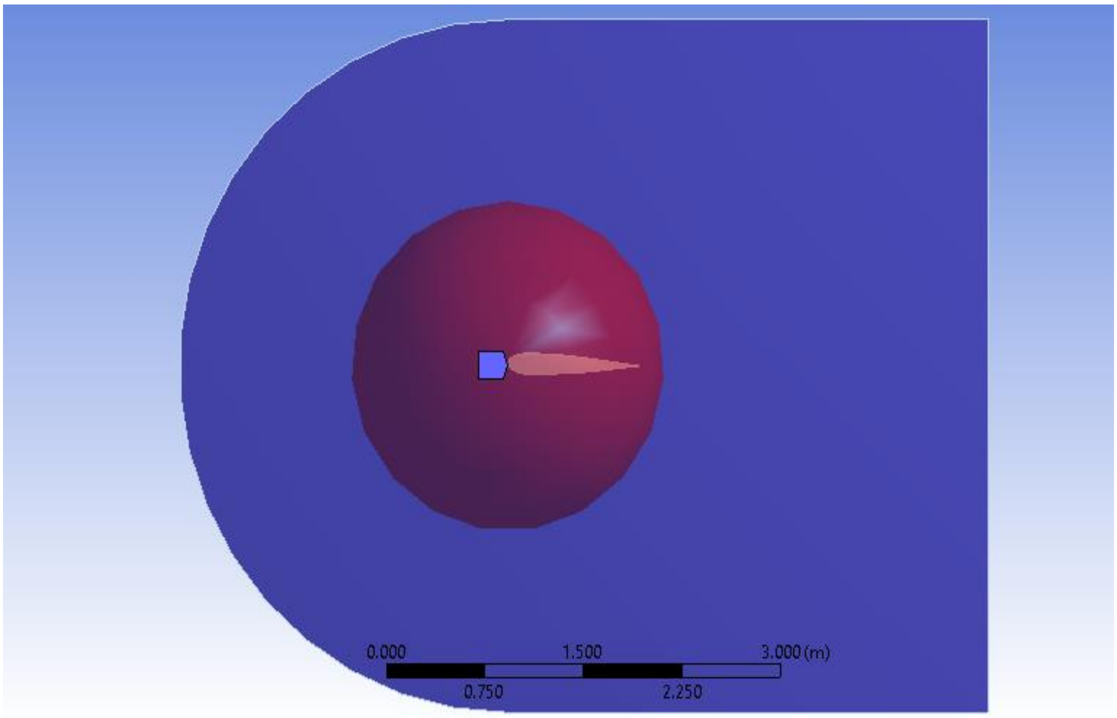
3 Inflation

4 Face structured meshing (if necessary)

Sphere of influence:

Sphere radius 1.2m

Sphere centre: global coordinate system



Body sizing of BOEING 737 airfoil

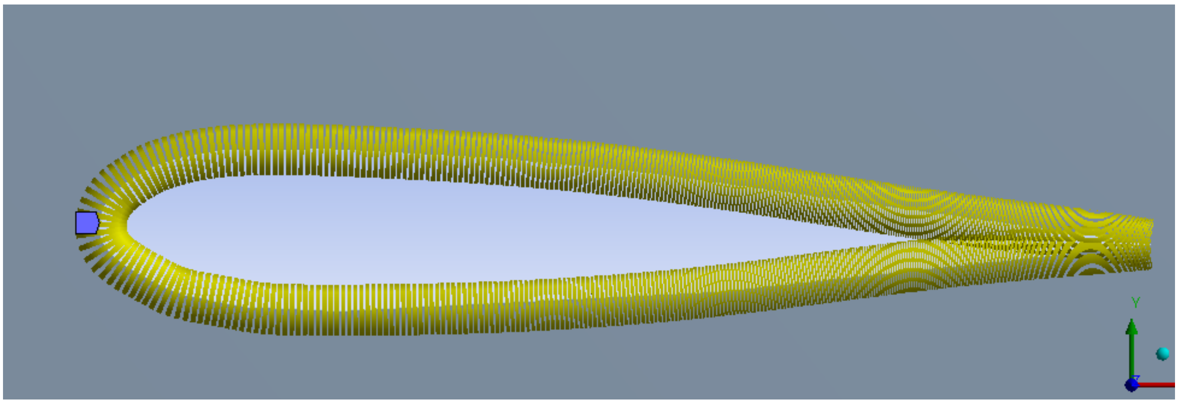
The sphere of influence creates a mesh in the vicinity of airfoil where the floe turns turbulent. it distinguishes the flow field from the wind tunnel setup

Edge sizing:

Element size ; 3e-003m

Behaviour : hard

Bias factor : defined 10



Edge sizing of an airfoil

This tool creates a well divided meshing elements around the perimeter of the airfoil. A bias factor can be included to identify the difference**.**

Inflation layer:

Boundary scoping method: geometry selection

Inflation: total thickness

Layers; 10 in number

Thickness(max): 6e-003m

Growth rate: 1.2

As it is observed inflation layers provide a more compacted layers of mesh elements around the boundary of airfoil (the black region around airfoil in fig)

DETAILS OF FINAL MESH

1. **SIZING**
   Advanced sizing: on proximity and curvature
   Relevance centre: fine (other options include Coarse and medium)
   Smoothing: high
   Proximity size sources: face and edge
   Min size: set to default(1.1582e-003m)
   Max face size: set to default(0.115820m)
   Max size: 0.50m

STATISTICS:

Depending upon the size of c mesh wind tunnel, the statistics differ.
C curve radius 2.5m
Rectangular length 3.7 m breadth 5m

No of nodes 16382

No of elements 15954

3.3.3 SETUP AND SOLUTION

In the previous sections, it has been established that a mesh needs to be perfectly generated for obtaining approximate solution. The methods which are followed to obtain the solution from a well-defined mesh are employed in **fluent** module of ANSYS.

PART 1 SETUP

Setup is the predecessor to obtaining a solution. In the fluent module, the software automatically reads the mesh data if its accurately produced.

Various solution techniques can be employed depending upon the solution we need. But, before that the geometrical/meshed entity must be totally transformed as a simulation of a wind tunnel apparatus.

So, the **first step** in the setup includes attributing each and every part of geometry.

Named selections: inlet, outlet, airfoil wall surface body entities are specified. This helps in understanding the process parameters from different levels.

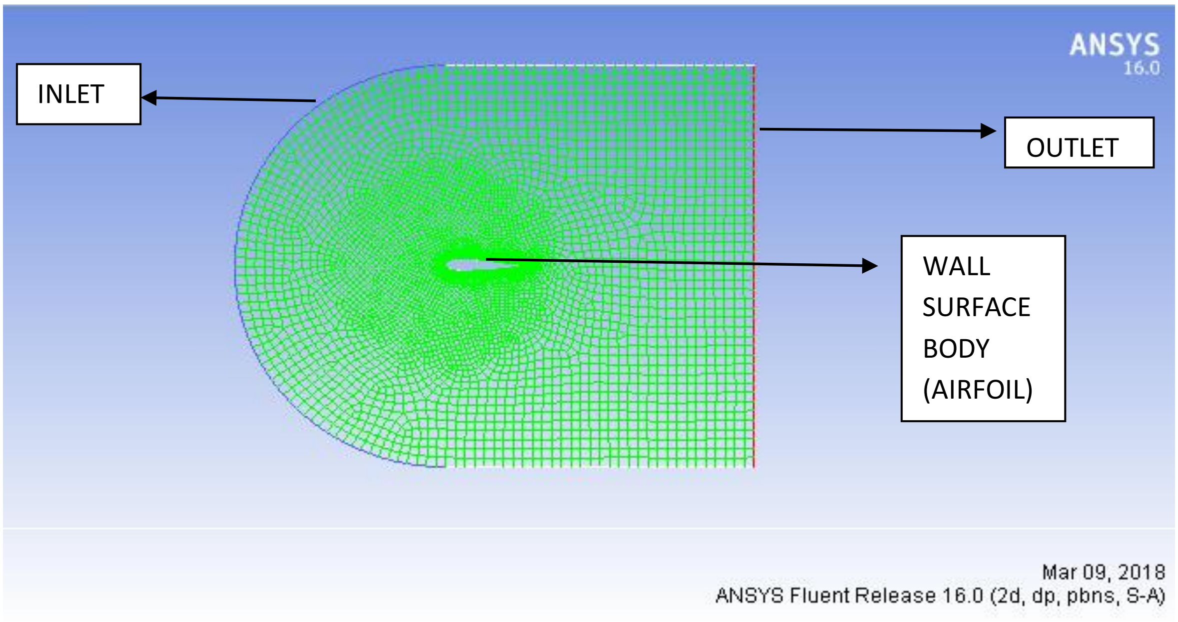


Fig SETUP WINDOW

Step 2: **solution type**: as mentioned earlier, different methods are available in the module. However for our requirement of flow over airfoil we choose

1. Pressure based/ absolute velocity model
2. Transient state
3. Viscous-spallart-allmaras(1 eqn) (solution formulations)
4. 2D planar space models

Above options are one of the numerous choices available for analysing flow over an airfoil. however solutions like lift, drag, pressure and velocity contours are available only in **viscous spallart allmaras type.**

Step 3: cell zone conditions this is an important selection which defines the tunnel boundary we have mentioned.

- Air flow is selected as the medium in this step. During analysis air flows past the airfoil
- Wall surface body of airfoil is taken as reference for flow

Step 4 : boundary conditions:

- The inlet is chosen as the entrance for airflow
- Following properties of fluid flow are considered by default
    - ➢ Type: velocity inlet
    - ➢ Velocity magnitude: 285.5 m/s (user defined)
    - ➢ Specification method: turbulent viscosity ratio
    - ➢ Turbulent viscosity ratio: set to 2 (highly turbulent flow)
    - ➢ Operating pressures: 122372 pa
    - ➢ Y-gravity: -9.8m/s^2
- The above boundary conditions vary for different airfoil designs and their cruising ranges and stall angles
- Dynamic mesh is employed when dealing with rotating angles of attack

Step 5 : Reference conditions

- The calculations are done taking inlet of the wind tunnel as the entrance

- Viscosity: 1.7894e-05 (default)
- Reference zone: wall surface body
- Temperature: 288.18 K
- Ratio of specific heats: 1.4
- Area of span: $1m^2$
- The span actually cannot be taken into consideration since the analysis is on 2D model. However, chord length is set at 1.12 m (approx)

PART 2 SOLUTION

After the inputs for analysis are set to required values, the next part is to select the required outputs we require. This comes under solution setup

In ANSYS, a console at the bottom right displays the actions taken and updates the conditions imposed. The results are also displayed in this console.

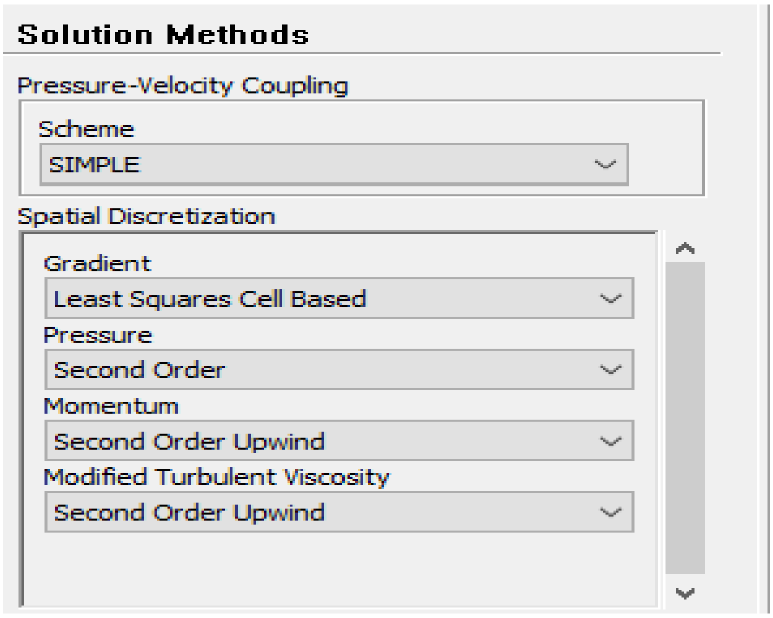
```
     parallel,
Done.

Preparing mesh for display...
Done.

Setting Post Processing and Surfaces information ...     Done.
Reading "\"| gunzip -c \"C:\Users\bharat surya\Desktop\boeing_files\dp0\FFF\Fluent\FF
Parallel variables...
Done.

  iter  continuity  x-velocity  y-velocity         nut        Cl-1        Cd-1     ti
!  291 solution is converged
   291  9.8640e-04  8.8911e-06  5.6444e-06  8.9675e-05  4.6363e-01  2.2592e-01  0:00:

 reversed flow in 7 faces on pressure-outlet 6.
!  292 solution is converged

 turbulent viscosity limited to viscosity ratio of 1.000000e+05 in 268 cells
   292  9.5461e-04  8.6990e-06  5.2788e-06  8.8454e-05  4.6322e-01  2.2593e-01  0:00:

  iter  continuity  x-velocity  y-velocity         nut        Cl-1        Cd-1     ti
   292  9.5461e-04  8.6990e-06  5.2788e-06  8.8454e-05  4.6322e-01  2.2593e-01  0:00:

 turbulent viscosity limited to viscosity ratio of 1.000000e+05 in 268 cells
```

console

Step 1: Solution methods

a) Pressure velocity coupling: SIMPLE(default)

b) Spatial discretization: The calculation of various entities like momentum, gradients, pressure come under this part.



SOLUTION METHODS

Step 2: monitors

The required results are graphically plotted in real time when we run the analysis. So monitors constitute those entities which form a part of our solution.

In our case, monitors include:

- Coefficient of lift
- Coefficient of drag
- Scaled residuals

These are plotted over a course of iterations and the final value can be printed to the console. The final converging value can be the average over iterations or the average over a course of time among the iterations where the graph is consistent

Step 3: the solution is initialized and checked for errors (if any).

A pop up window specifies the error which has to be modified

Step 4: to run the calculation, the software takes a number of user specified iterations to converge to the solution. More the number of iterations, more the solution convergence. However, the limitation here is that the software tends to slow down since the cpu, gpu usage increases, and some systems may not be able to handle the processing speed. Therefore, iterations are specified around 300-500.

Also, after a number of iterations less than the specified, the solution converges to its optimum. Hence the remaining iterations are redundant and only leads to increase in processing time.

The plots which we obtain on monitor are:

- Cd (drag coefficient) vs iterations
- Cl (lift coefficient) vs iterations

The console prints the converging value of Cl and Cd

Also in the final stage, ANSYS has the provision to plot filled contours of

- Pressure (static dynamic, absolute)
- Velocity magnitude (above and below)
- Lift coefficient over iterations
- Drag coefficients over iterations

These contours are specified and included for each of the airfoils NACA 0012, NACA2412, BOEING 737in the coming sections in detail.

CHAPTER 4

FLUENT ANALYSIS OF AIRFOILS

In this section, the pressure, velocity contours of meshed airfoil sections have been included along with the drag and lift approximations.

4.1 ANALYSIS OF BOEING 737- ROOT AIRFOIL

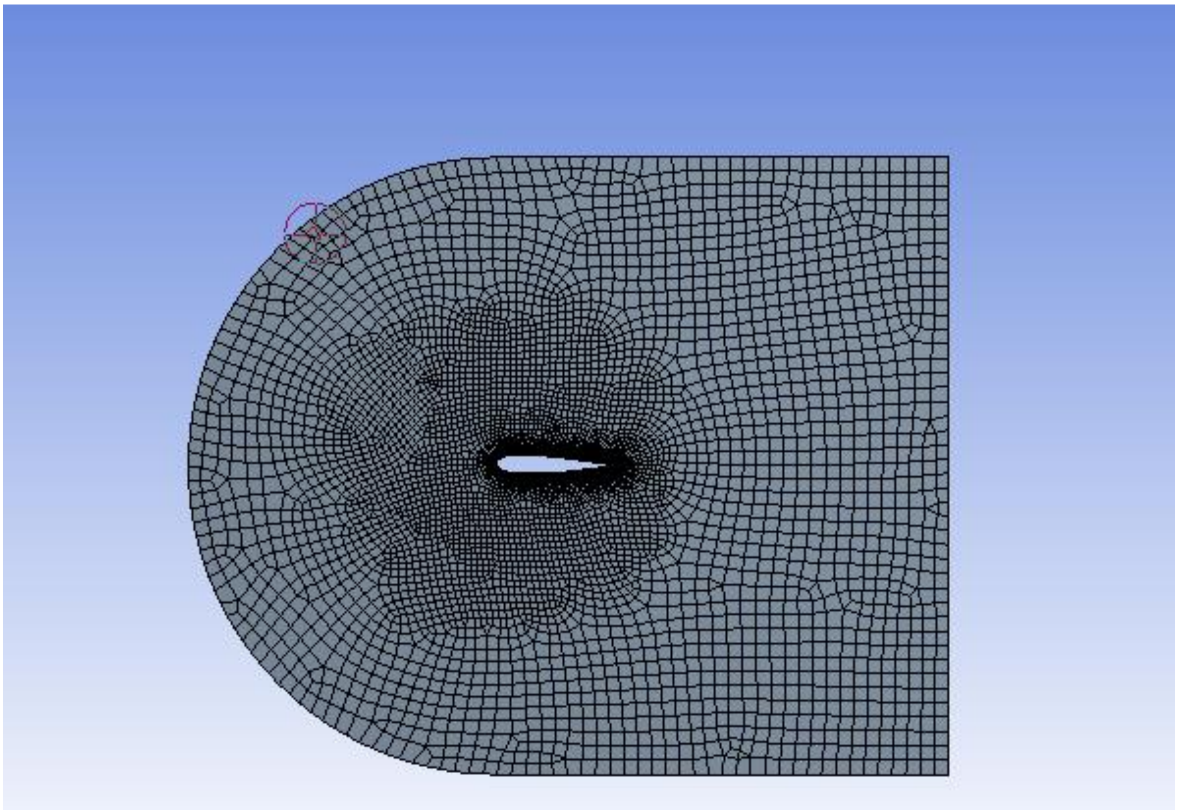


Fig 4.1 MESHED VIEW OF BOEING 737 AIRFOIL

The vicinity of the airfoil has been meshed with different layering techniques so that the flow distribution over the airfoil provides more accurate results during simulation.

Using transition SST – 4 eq model, and the same setup parameters mentioned in previous sections, following are the pressure and velocity contours at different attack angles.

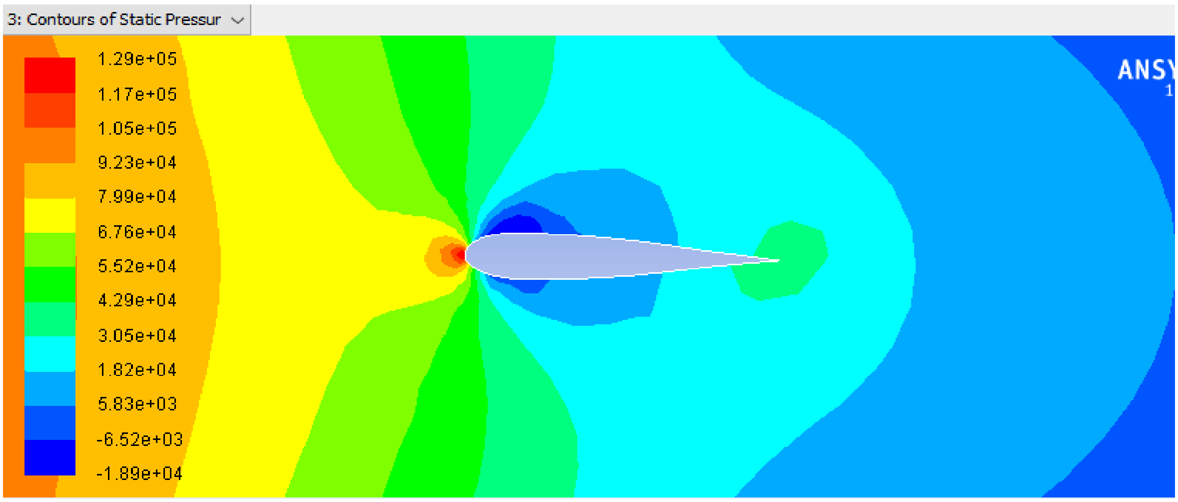


FIG 4.2 STATIC PRESSURE CONTOUR- BOEING 0^{O} AOA

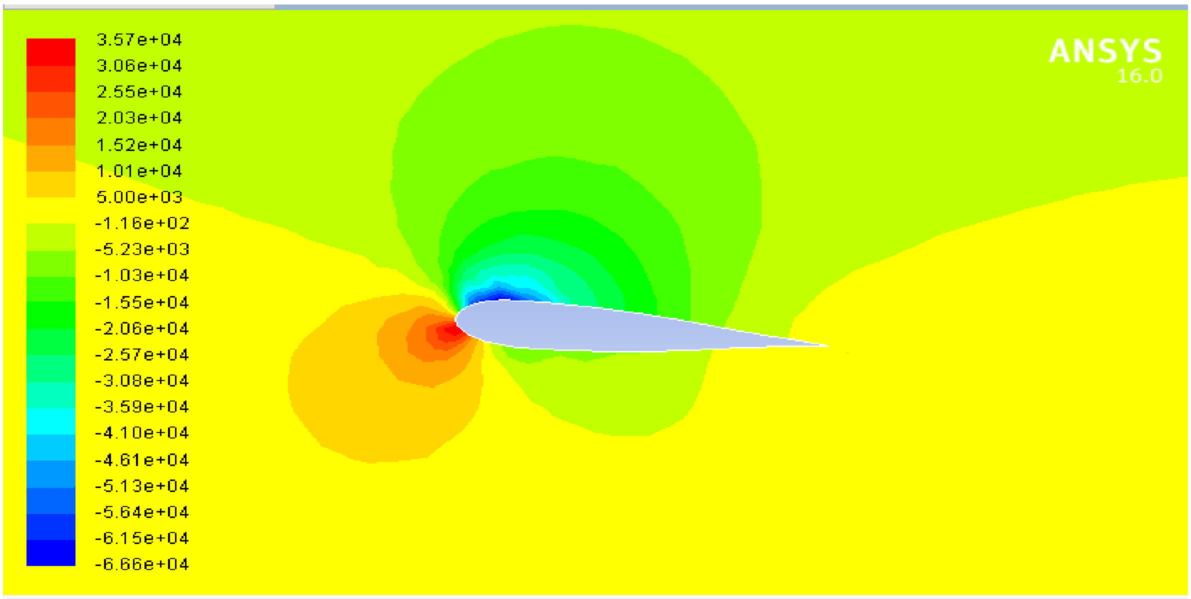


FIG 4.3 STATIC PRESSURE CONTOUR BOEING 4^{O} AOA

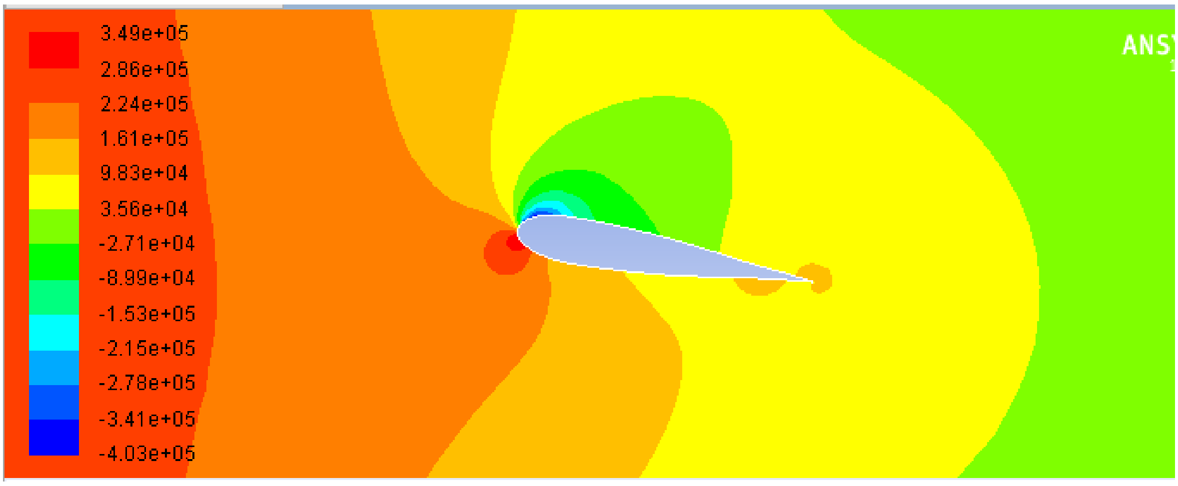


FIG 4.4 STATIC PRESSURE CONTOUR-BOEING 9^0 AOA

ANGLE OF ATTACK	MIN PRESSURE(Pa)	MAX PRESSURE(Pa)
0	-88035	346922.5
4	-66600.34	356831.96
9	-403478	349192.9

TABLE 4.1 PRESSURE GRADIENT AT VARYING AOA

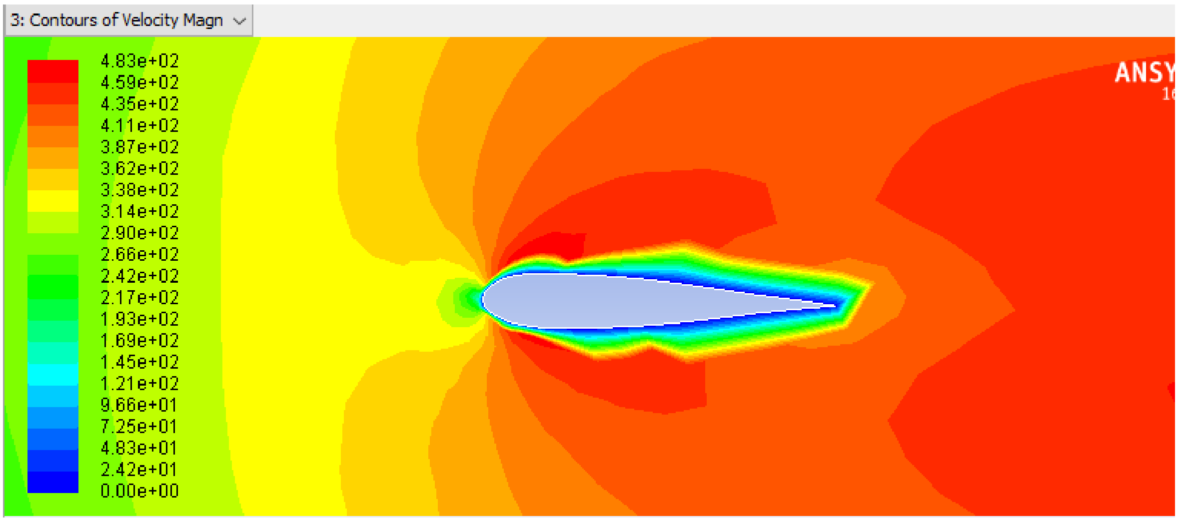


FIG 4.5 VELOCITY CONTOUR BOEING 0^O AOA

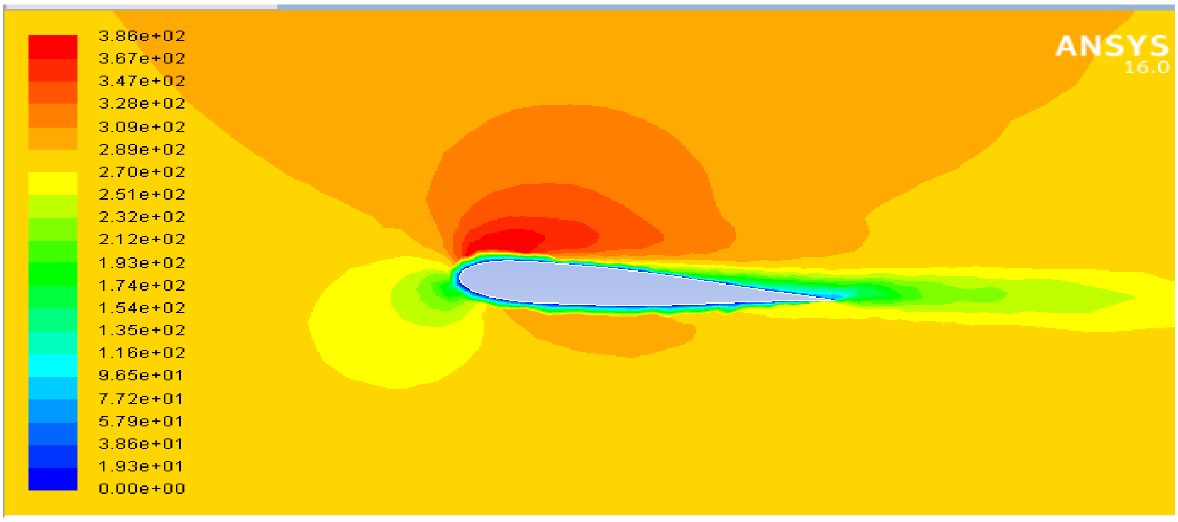


FIG 4.6 VELOCITY CONTOUR BOEING 4^{O} AOA

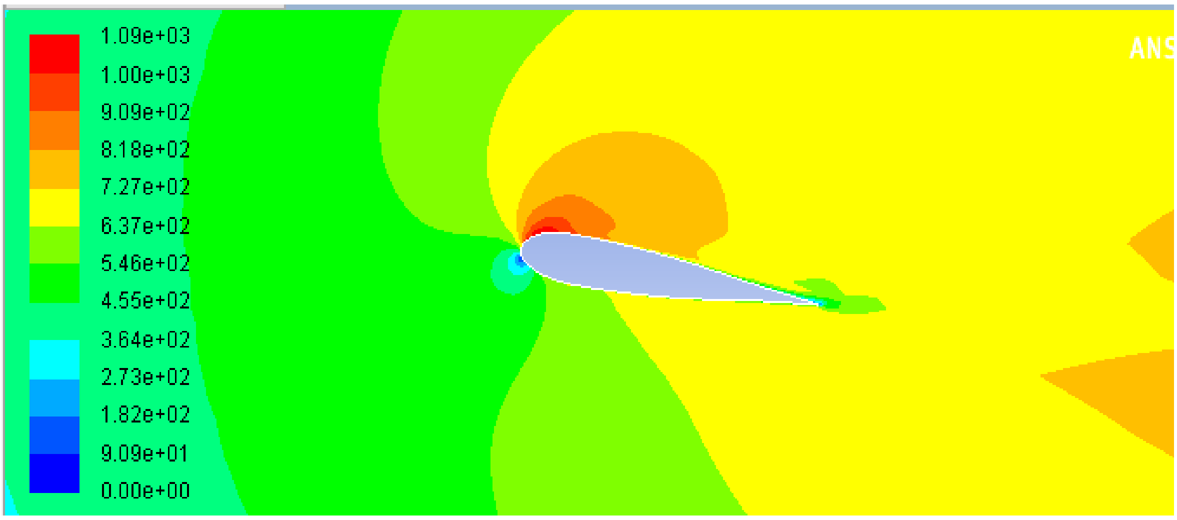


FIG 4.7 VELOCITY CONTOUR BOEING 9^{O} AOA

ANGLE OF ATTACK	VELOCITY GRADIENT (m/s)
0	483
4	386
9	1091

TABLE 4.2 VEOLCITY MAGNITUDES AT VARIOUS AOA

4.1.1 Analysis results:

Lift and drag coefficients:

- the lift and drag coefficients provide the necessary relationship between free stream velocity and lift and drag forces respectively.
- It is known that lift force should overcome the gravity and drag must be minimised. this drag coefficient is also called skin friction coefficient.

$$C_L = \frac{2L}{\rho V^2 A}$$

Where cl- lift coefficient, v-velocity of flow, A- surface area

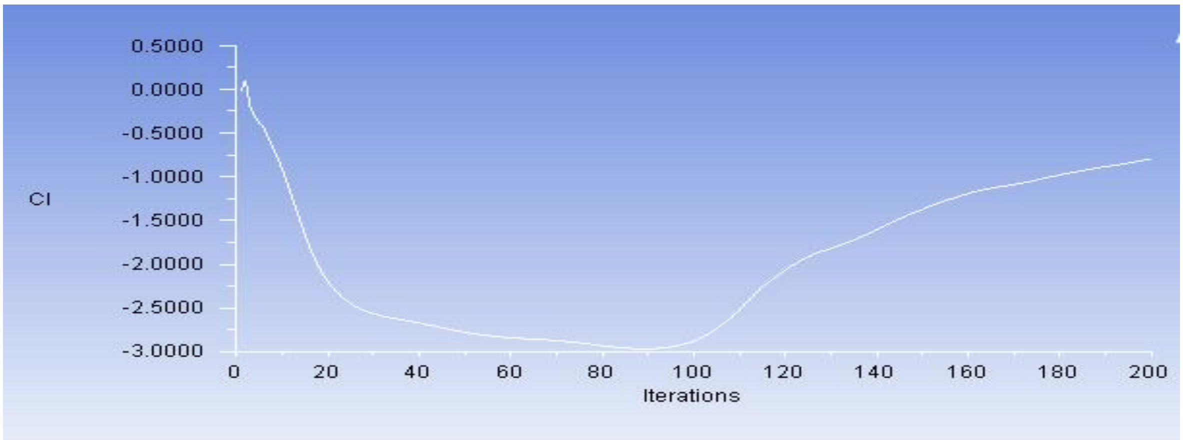
- The lift coefficient is proportional to lift force generated.

From the simulations carried out at different angles of attack, the lift and drag coefficients have been found. These are provided in the following table.

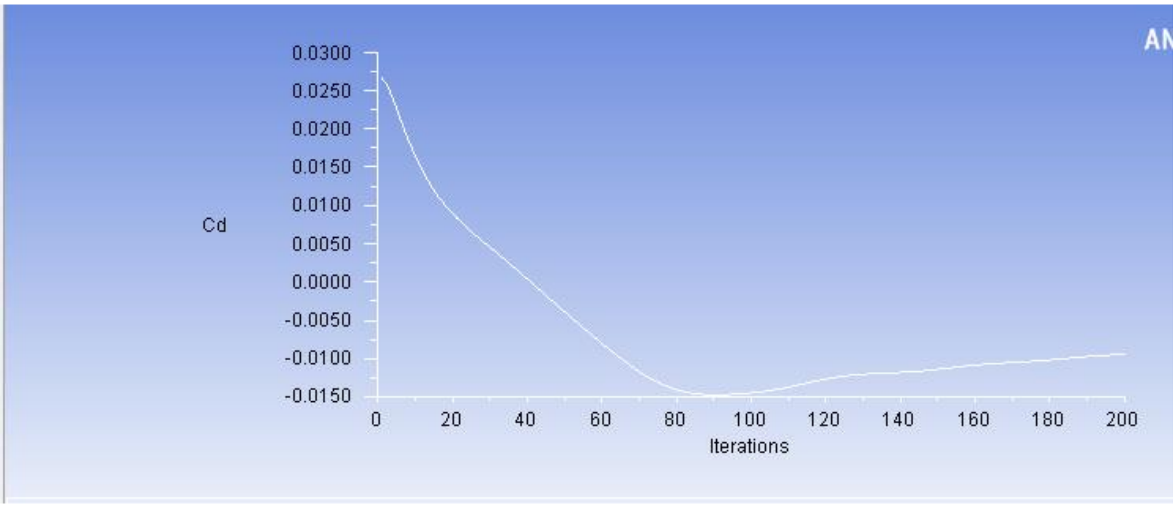
Angle of attack	Cl (lift coefficient)	Cd (drag coefficient)
-2	-0.095	0.015
0	0.254	0.068
4	0.458	0.043
6	0.621	0.012
9	0.7125	0.009

Table 4.3 cl and cd at different AOA

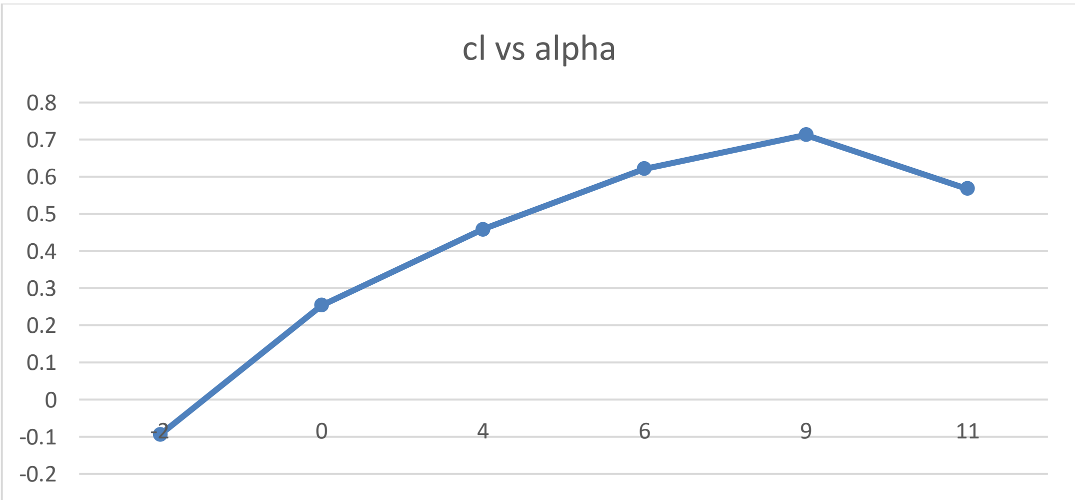
The cl/cd ratio is found to be maximum at AOA 0f 9^0. Hence , it is the stall angle of attack.



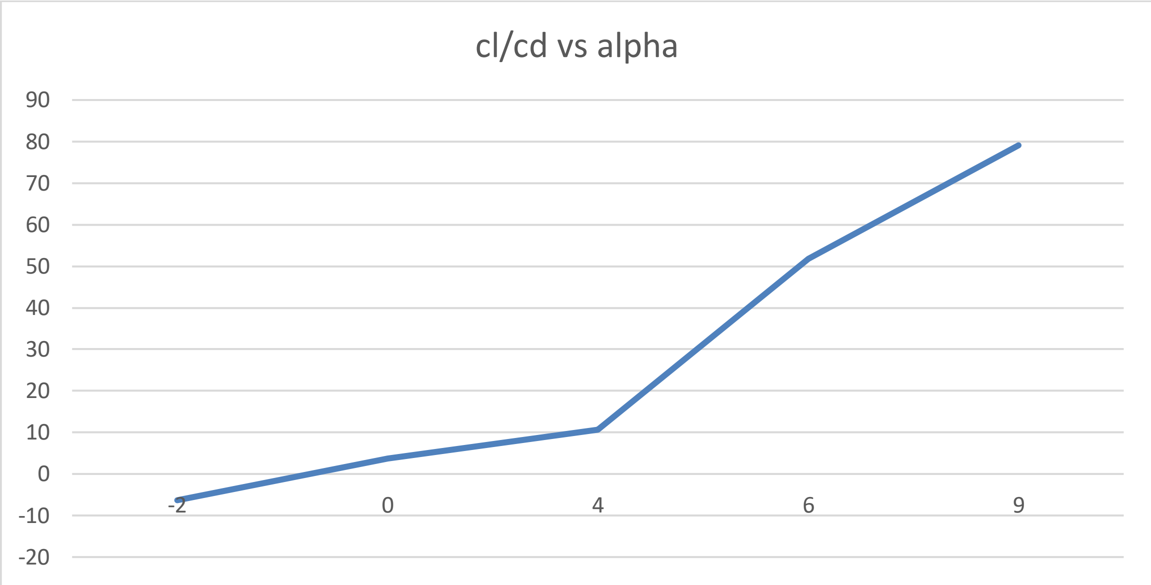
Graph 4.1 cl vs iterations (200) at stall angle of attack (9^o)



Graph 4.2 cd vs iterations at stall angle of attack (9^0)



Graph 4.3 cl vs alpha (AOA)



Graph 4.4 cl/cd vs alpha(A0A)

In the global coordinate system of airfoil modelling i.e.; in XY plane,

Lift force Polar X Y Z corresponds to 0 1 NA

Drag force polar X Y Z corresponds to 1 0 NA

Angle of attack	Lift force generated(N)
0	25400
4	22441
9	156248

Table 4.4 Lift force at various AOA

Angle of attack	Drag force
0	11013
4	2140
9	21172

Table 4.5 drag force at various AOA

4.2 ANALYSIS OF NACA 0012 SYMMETRICAL AIRFOIL

Symmetrical airfoils such as NACA 0012 are not used at high Mach numbers since they cannot provide sufficient lift. Hence their analysis is limited to maximum flow velocity of 80m/s.The analysis has been carried out at three different angles of attack (alpha) and the lift and drag coefficients have been caluculated.

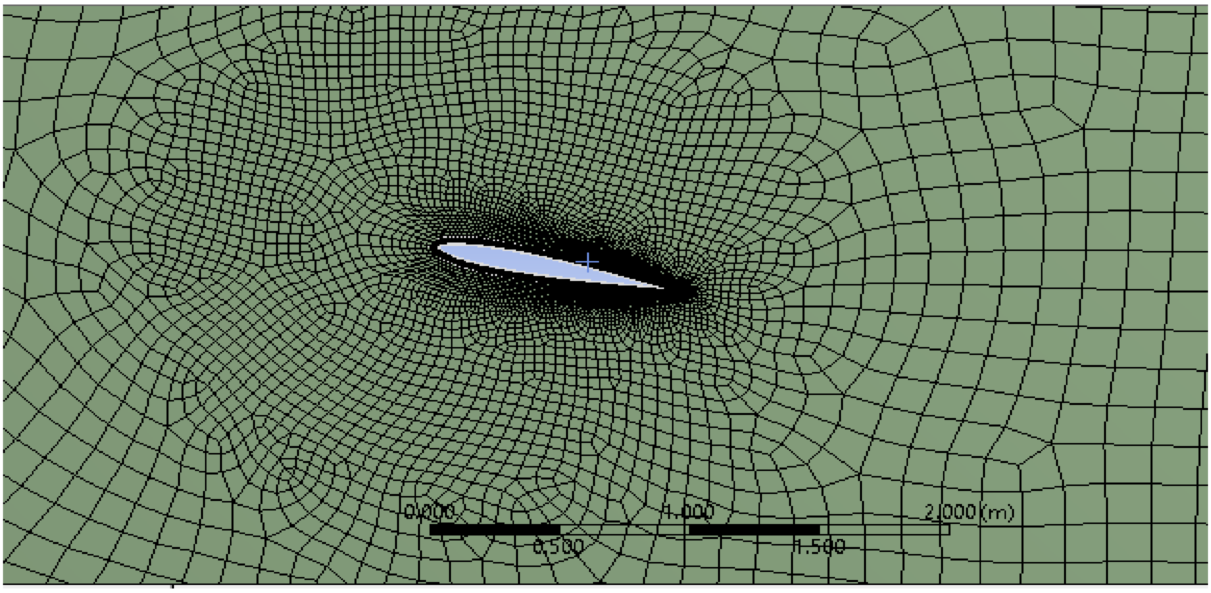


Fig 4.8 Generated mesh of NACA 0012 symmetrical airfoil

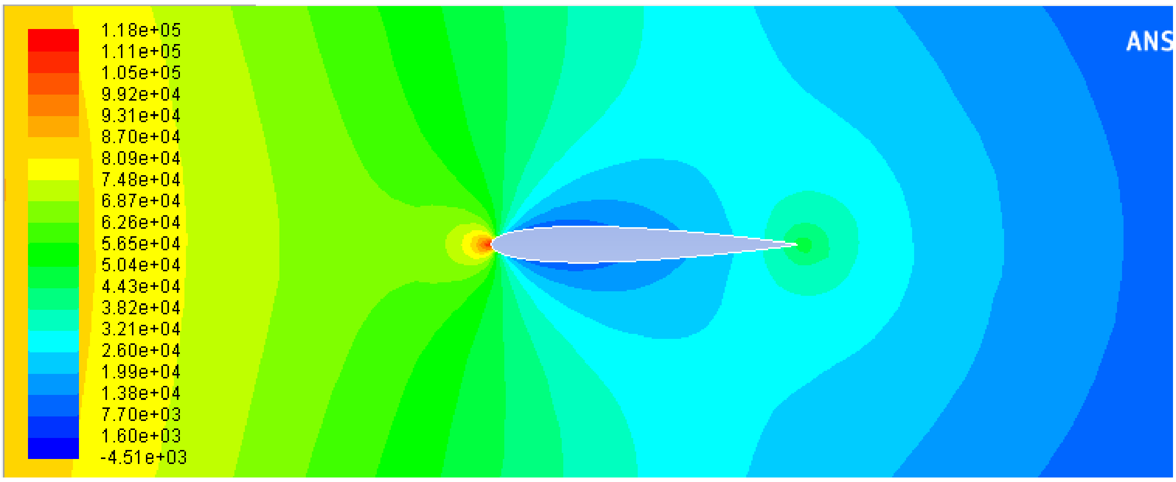


Fig 4.9 Pressure contour of NACA 0012 at 0^0 AOA

From, the pressure contour, it can be inferred that since the airfoil being a symmetrical profile, the pressure distribution along the curve remains the same. Such symmetrical pressure distributions cannot generate lift. Hence various angles of attack are needed to be employed for generation sufficient lift.

Below are some other profiles which show a sufficient pressure variance.

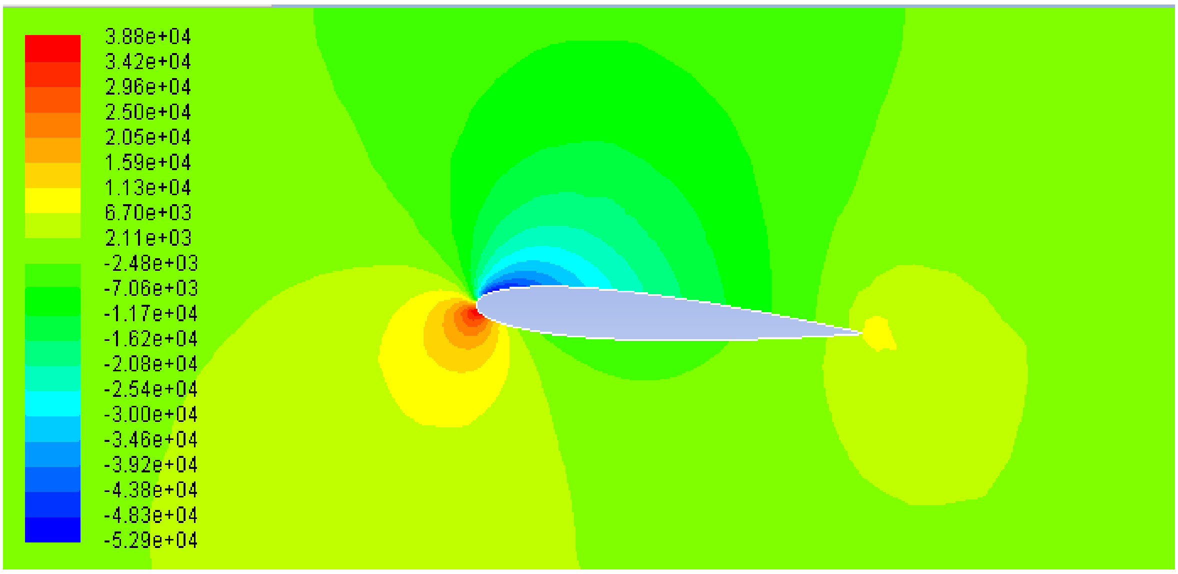


Fig 4.10 pressure contour at 4^0 AOA

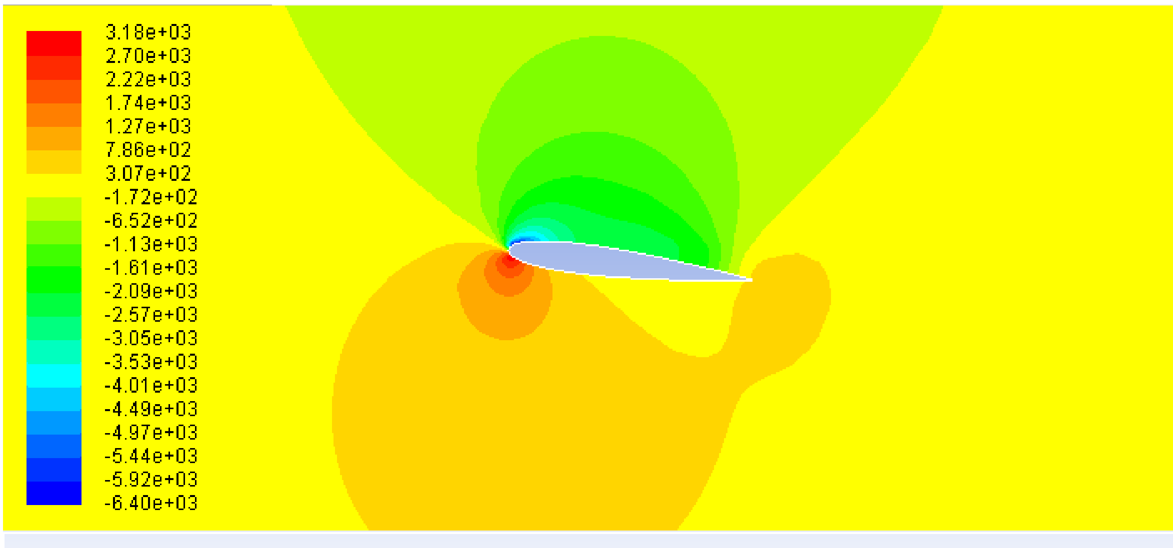


Fig 4.11 pressure contour at stall AOA 7^0

Angle of attack	Min pressure (Pa)	Max pressure(Pa)
0	-4506	1175
4	-5239	3880
7	-6403	3182
12	-4059	2558

Table 4.6 Pressure gradient at various AOA

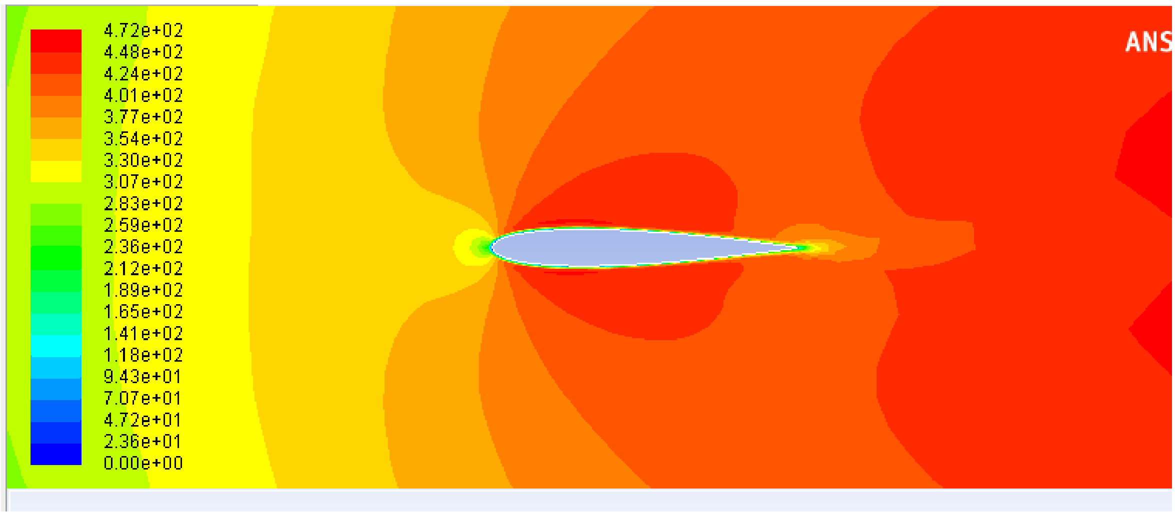


Fig 4.12 velocity contours at 0^0 AOA

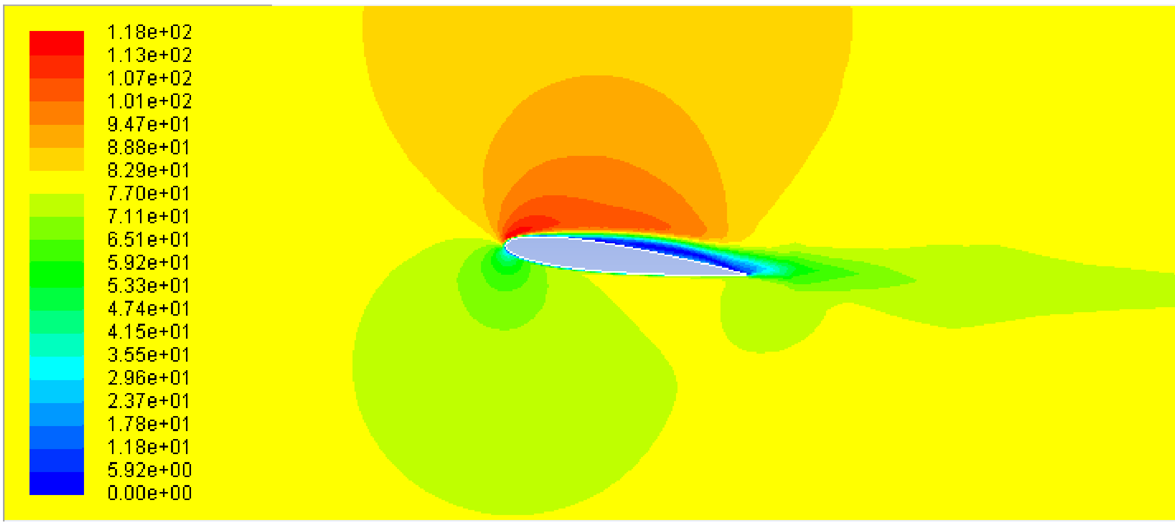


Fig 4.13 velocity contours at stall AOA 7^0

4.2.1 ANALYSIS RESULTS

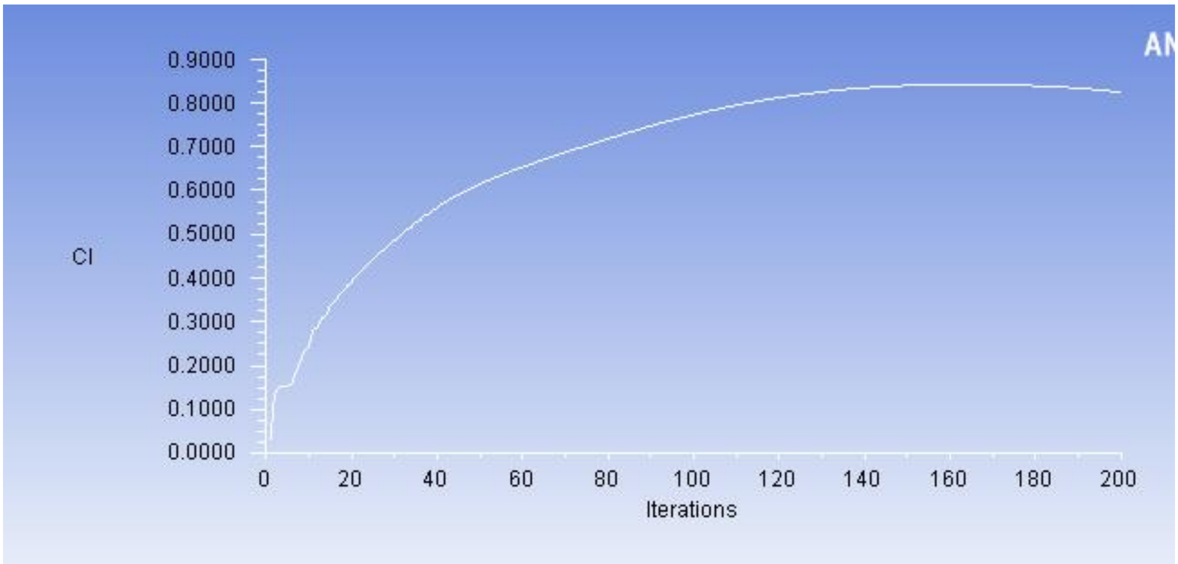
Lift and drag coefficients:

Symmetrical airfoils do not exhibit pressure and velocity gradient at the neutral positon, hence the drag and lift coefficients are equal to zero, however the approximation depends upon the mesh generated.

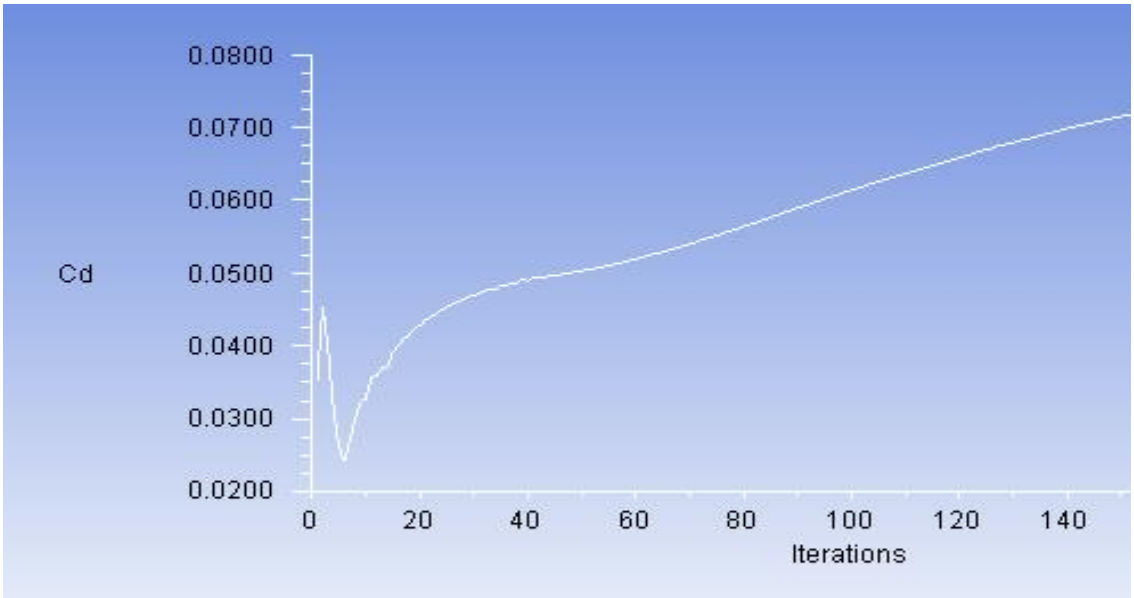
From our analysis the stall angle has been found out to be at 7 ^{0} to the XY plane of analysis.

Angle of attack	Lift coefficient (cl)	Drag coefficient(cd)
0	- 0.0017	0.178
4	0.2	0.13
7	**0.8**	**0.05**
12	0.5	0.1

Table 4.7 cl, cd at different AOA

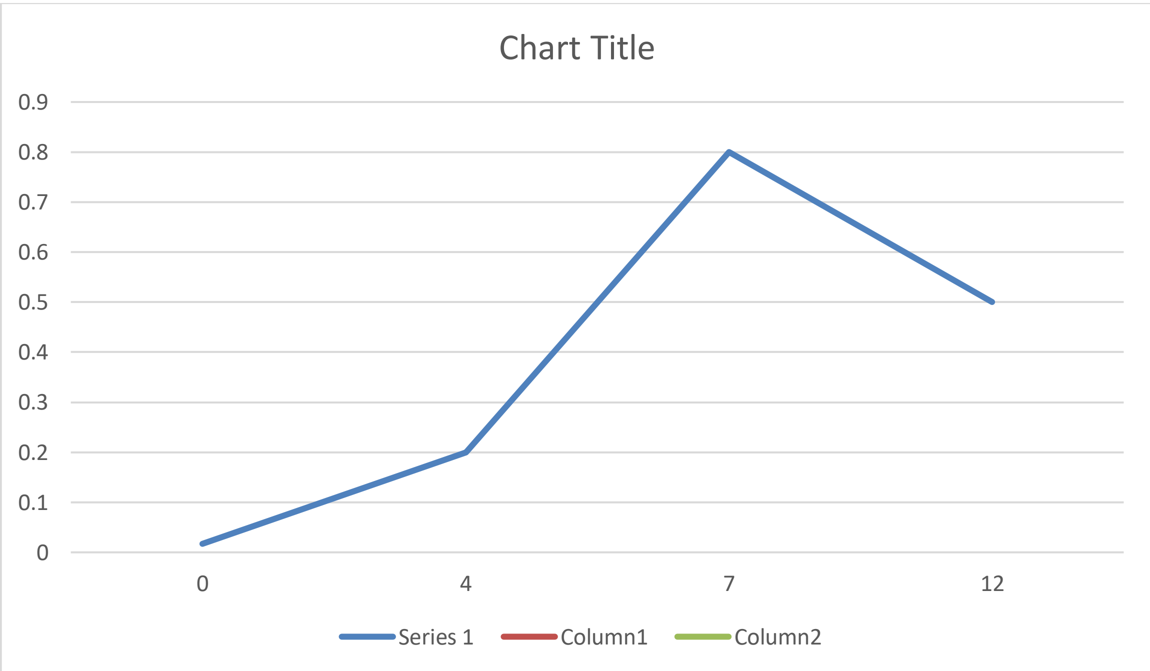


Graph 4.5 cl vs iterations at stall angle of attack 7^0



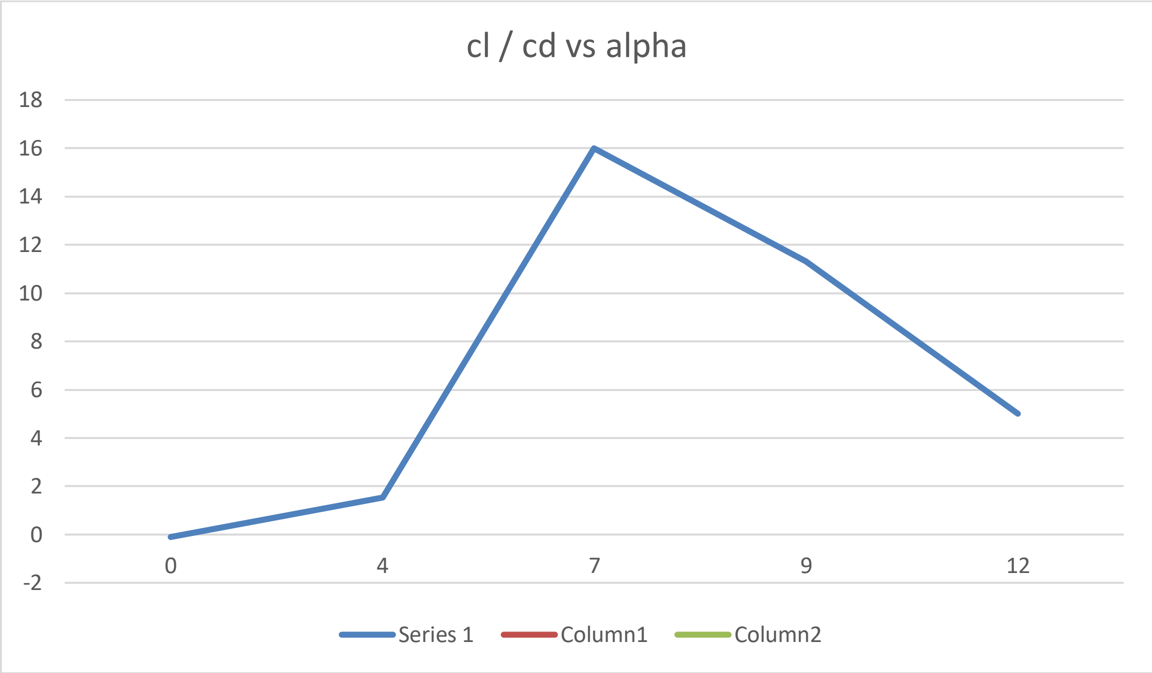
Graph 4.6 cd vs iterations at stall angle of attack 7^0

Iterative approach converges the solution more accurately. 200 measures have been taken into consideration for finding out the lift and drag coefficients.



Graph 4.6 cl vs AOA

The graph indicates that after the stall AOA, lift coefficient decreases with increase in AOA.



Graph 4.7 cl/cd vs AOA of NACA 0012

It can be found that cl/cd ratio at a mere laminar flow is higher at 7 degrees for NACA 0012.

Drag and lift forces:

since, we have been dealing with airfoil models of scaled width and mass, the forces obtain are at a scale of what actually there are. These airfoils are not put into actual use as these are intermediate design obtained in developing a full scale model.

Angle of attack	Lift force	Drag force
0	-349	337.4
4	3145	2676
7	**2907**	**273**
12	1694	525

Table 4.8 drag and lift forces at various AOA

4.3 ANALYSIS OF NACA 2412 AIRFOIL

The analysis is carried out at velocities of 80m/s. these airfoils exhibit greater lift and drag at very less Mach numbers, i.e; at low velocity flows. However our analysis is to investigate its performance where turbulence cannot be neglected.

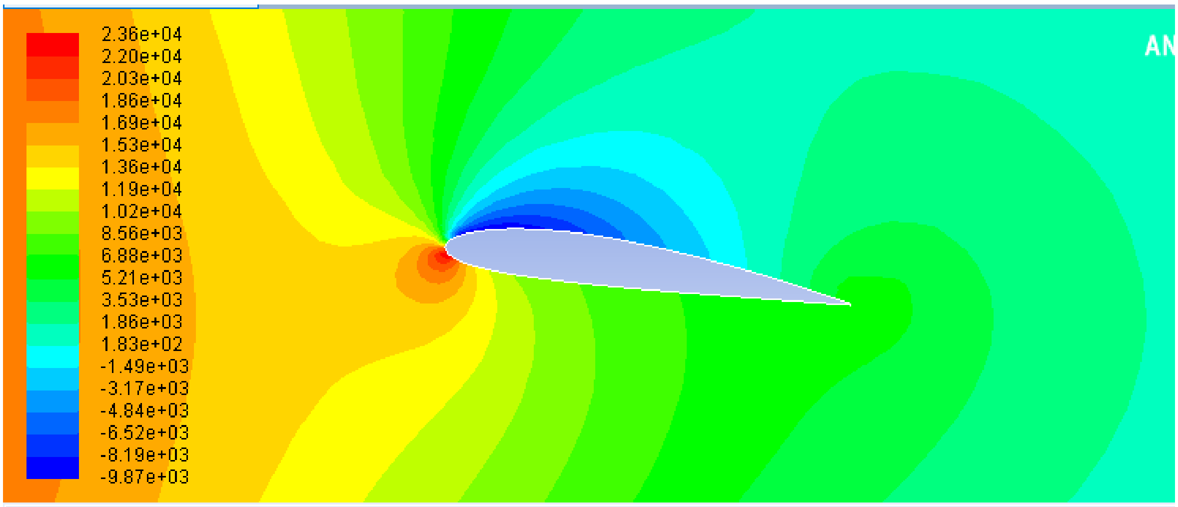


Fig 4.14 pressure contour at AOA 8^0

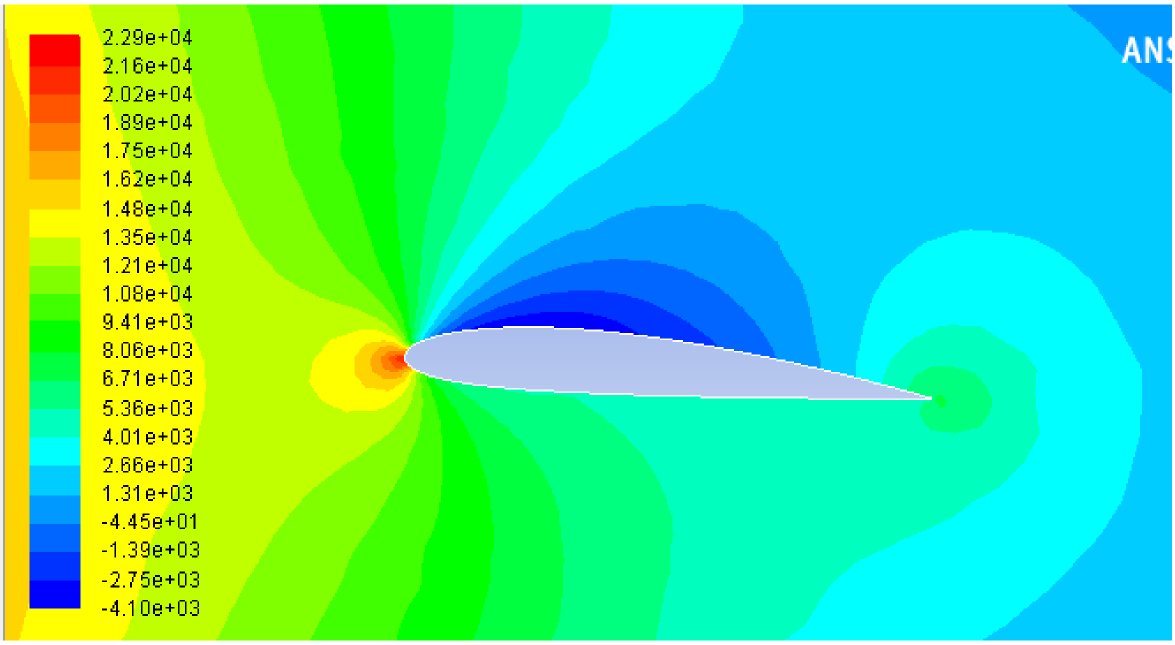


Fig 4.14 pressure contour of NACA 2412 at stall AOA 4^0

Angle of attack	Min pressure	Max pressure (Pa)
0	-1152	17854
4	-4095	22912
8	-9869	23638
10	-8541	19258

Table 4.9 pressure gradient at various AOA

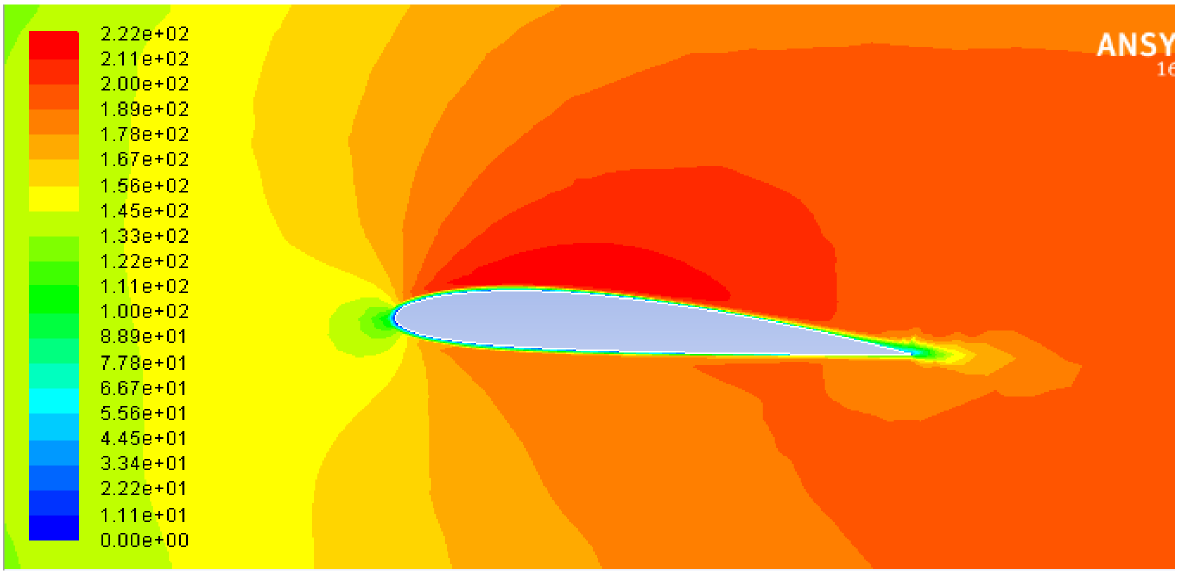


Fig 4.15 velocity contour at stall AOA 4 ^{0}

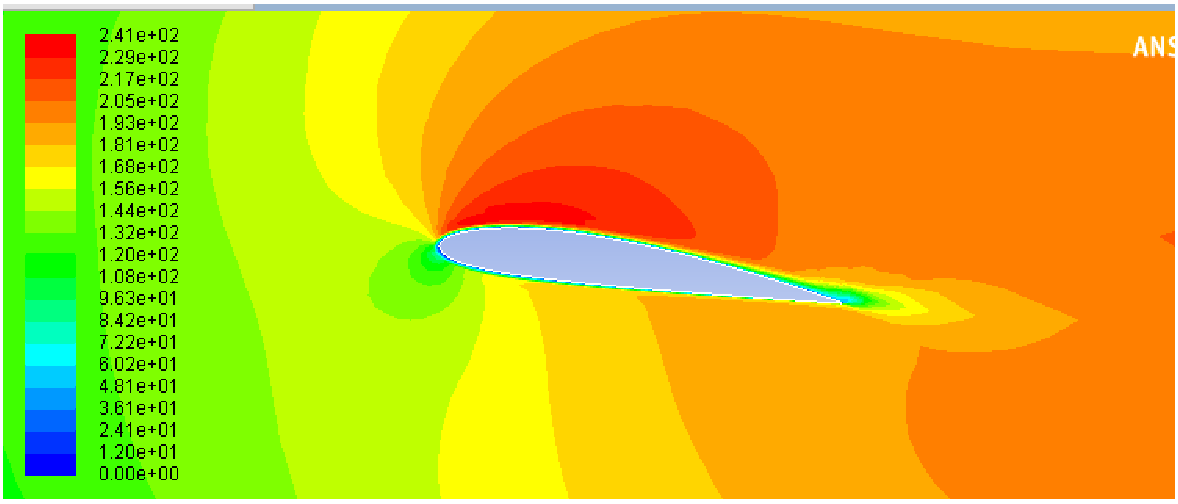


Fig 4.16 Velocity counter at AOA 8^{0}

Angle of attack	Maximum velocity
0	214
4	222
8	240

Table 4.10 velocity magnitude at various AOA

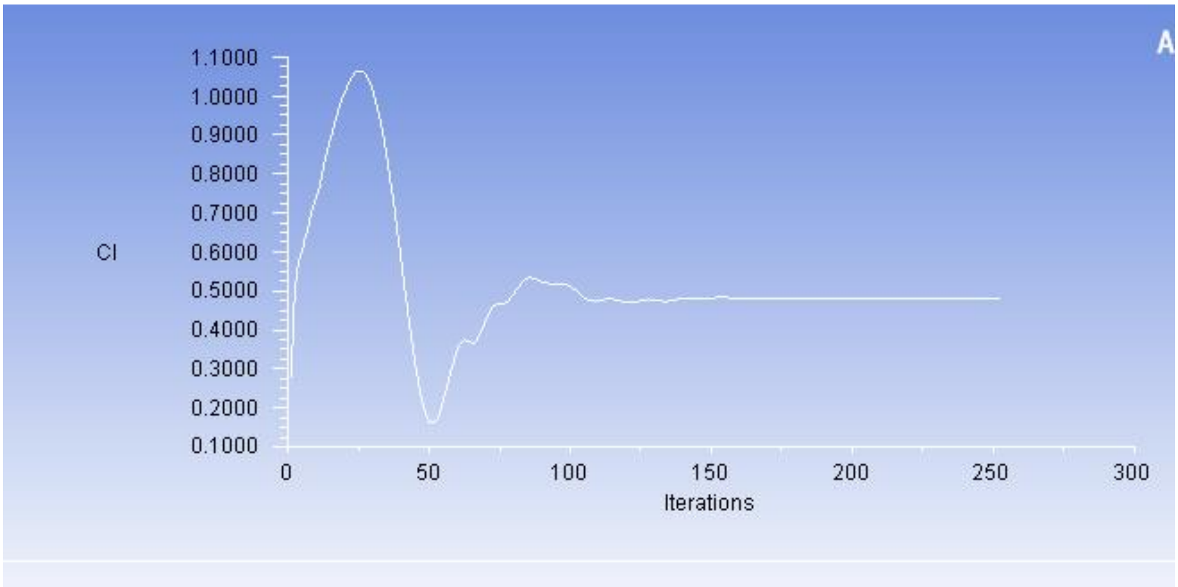
4.3.1 ANALYSIS RESULTS

Lift and drag coefficients:

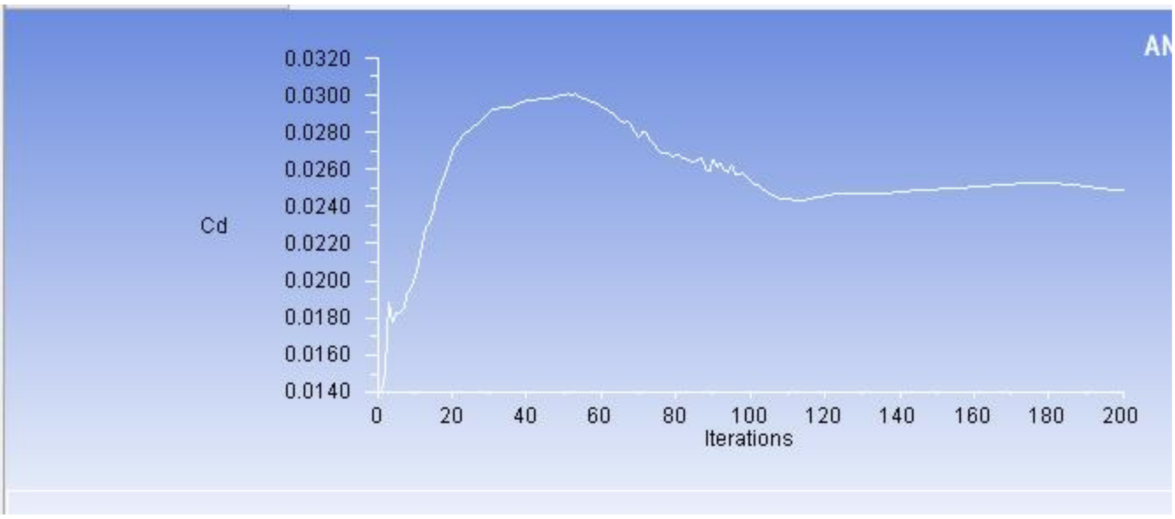
As mentioned. NACA 2412,being asymmetrical have higher lift and drag coefficients at low velocities of flow. The lift and drag coefficients over a number of iterations are given below.

Angle of attack	Lift coefficient (cl)	Drag coefficient (cd)
0	0.6	0.4
4	**0.25**	**0.026**
8	0.28	0.03
10	0.6	0.18

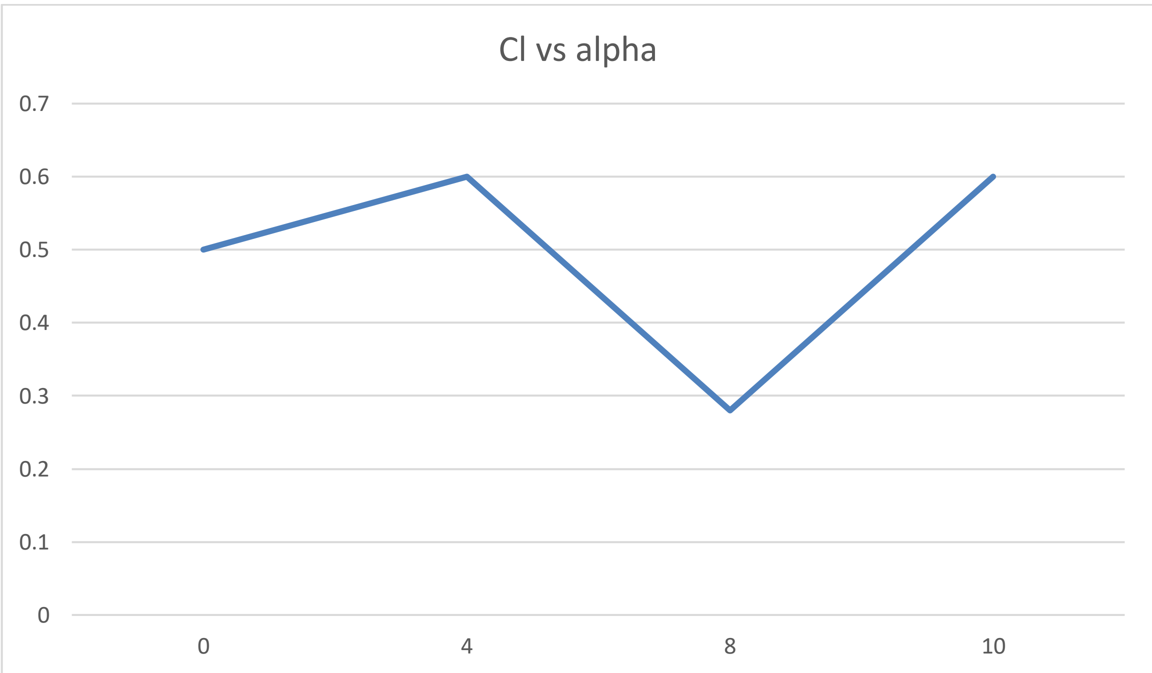
Table 4.11 lift and drag at various AOA



Graph 4.8 cl vs iterations at stall AOA 4^0



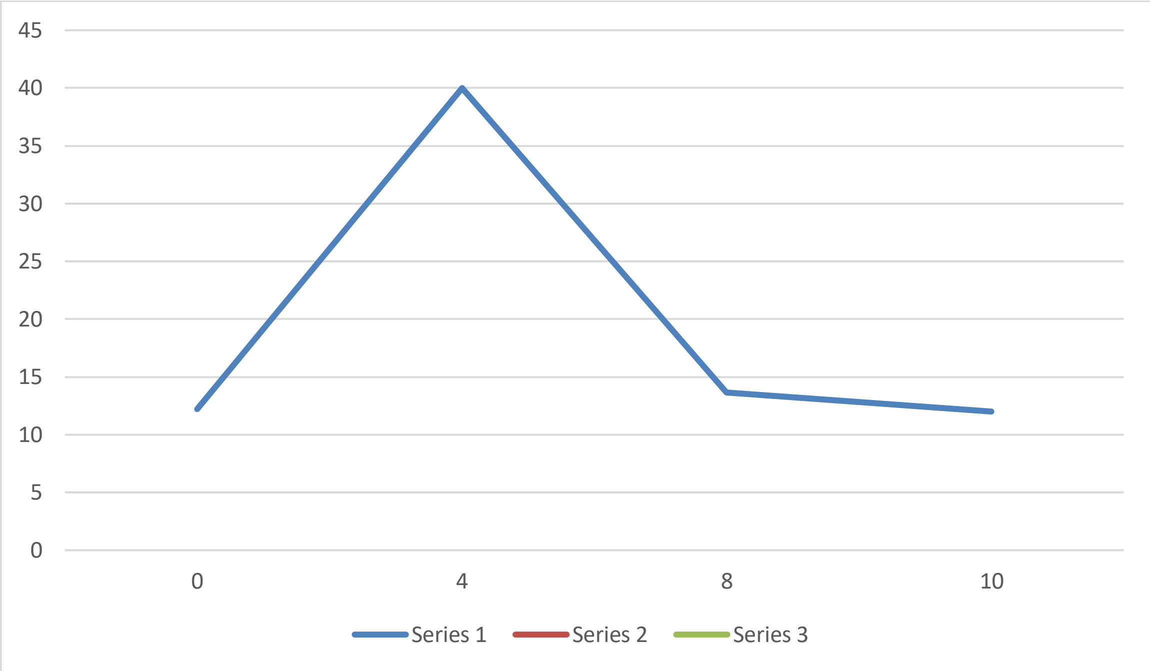
Graph 4.9 Cd vs iterations at stall AOA



Graph 4.10 cl vs alpha

The graph indicates a subtle explanation of variation of lift and drag coefficients with angle of attack. Therefore, these designs are rather confined to experimental purposes only.

However, the airfoil exhibited a good lift at angle of attack of 4 degrees. Therefore, it can be considered as the stall angle of attack.



Graph 4.11 cl/cd vs alpha (AOA)

Lift and drag forces

Below is the variation of lift and drag forces with Angle of attack.

Angle of attack	Lift force	Drag force
0	28714	19857.2
4	59350	1237
8	11644	1835

Table 4.12 lift and drag forces at various AOA

CHAPTER 5

RESULTS VALIDATION AND CONCLUSION

The analysis of airfoils has been done subjected to various operating parameters, and the results obtained have been included. The next step in any analysis is to validate the results with some reference and then employ it into actual design

In case of ANSYS analysis, the data obtained from the module can be validated in following ways:

1) Experimental test results that include wind tunnel setup
2) Mathematical convergence
3) Source data validation

In our context, resources being limited, source data validation has been employed. The inputs have been provided from the database of airfoils which also gives the Xfoil predictions of the airfoils in graphical format. These values can be compared to our analysis results.

5.1 Analysis results:

- Importance has been given to finding lift and drag coefficients of various airfoils
- Pressure and velocity contours did show necessary gradient required that tallies with the lift forces generated
- Stall angle of attack where an airfoil exhibits satisfactory lift is considered to be the prime objective of the analysis
- Comparing these values with Xfoil predictions might differ in values since it depends on the accuracy of meshing and setup.
- The results , we obtained are presented in tabular format.

airfoil	Stall angle of attack
BOEING 737	9^0
NACA 0012	7^0
NACA 2412	4^0

Table 5.1 Stall AOA

Coefficient of lift and drag are taken at stall angle of attack and they turn out to be:

airfoil	Cl (stall)	cd
BOEING 737	0.7125	0.009
NACA 0012	0.8	0.05
NACA 2412	0.6	0.009

Table 5.2 cl and cd at stall AOA

The lift and drag forces mentioned in the previous sections are mere calculations of a massless Boolean operation applied over a surface. The actual lift and drag forces can be estimated when the analysis is over 3dimebsional wing. However, a 3 dimensional analysis more complex and hectic process.

5.2 VALIDATION OF NACA 0012 WITH XFOIL DATA:

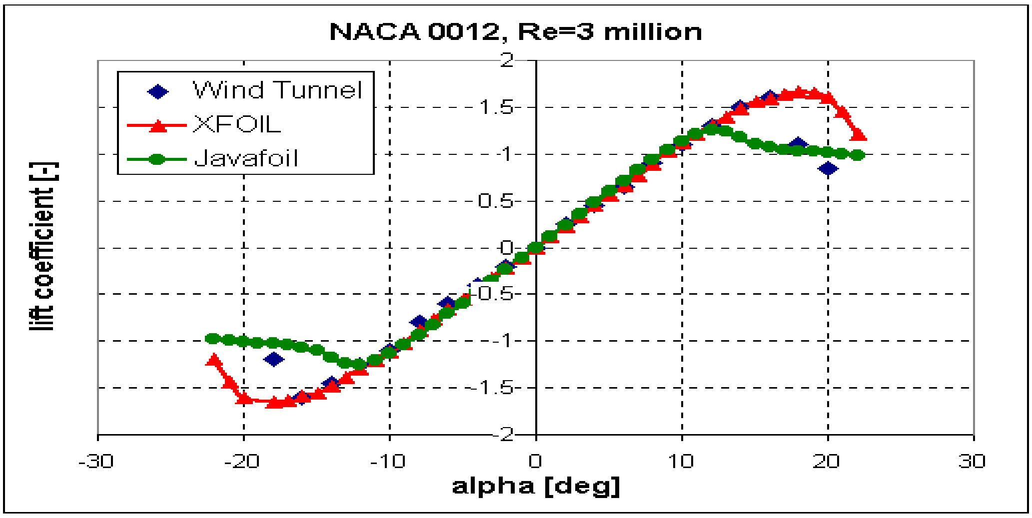
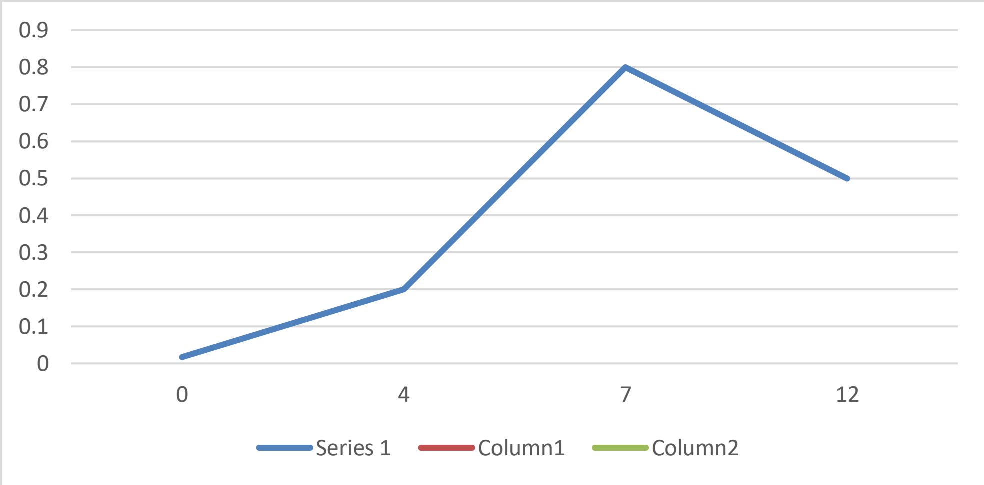


Fig 5.1 XFOIL plot of alpha vs lift coefficient.

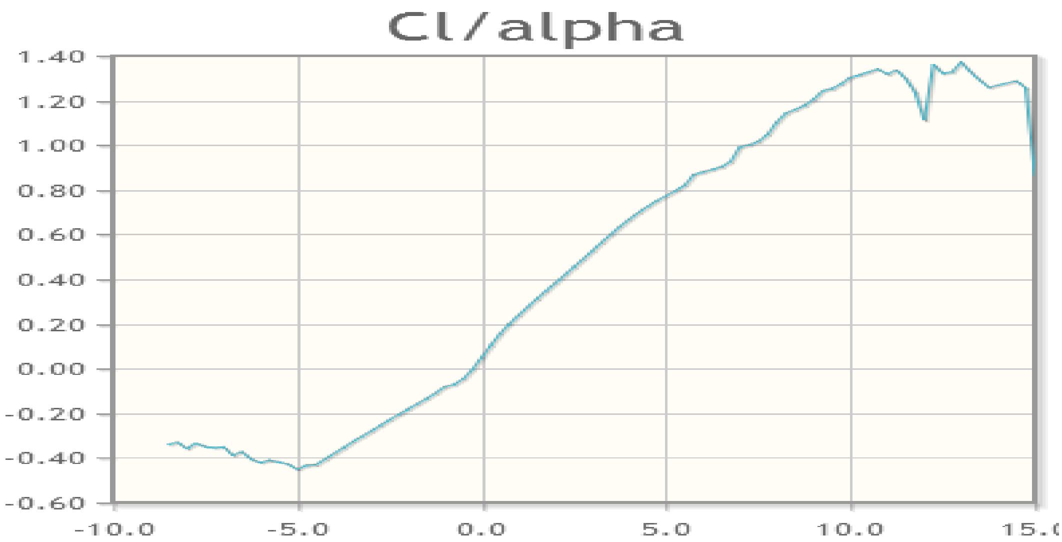


Graph 5.1 cl vs alpha analysis results

Inference:

- The analysis plot shows a cd of 0.8 at 7 degrees.
- The Xfoil data also shows similar range of lift coefficient.
- However, these plots are at simulations carried out at low velocities, so compressibility effects are neglected.

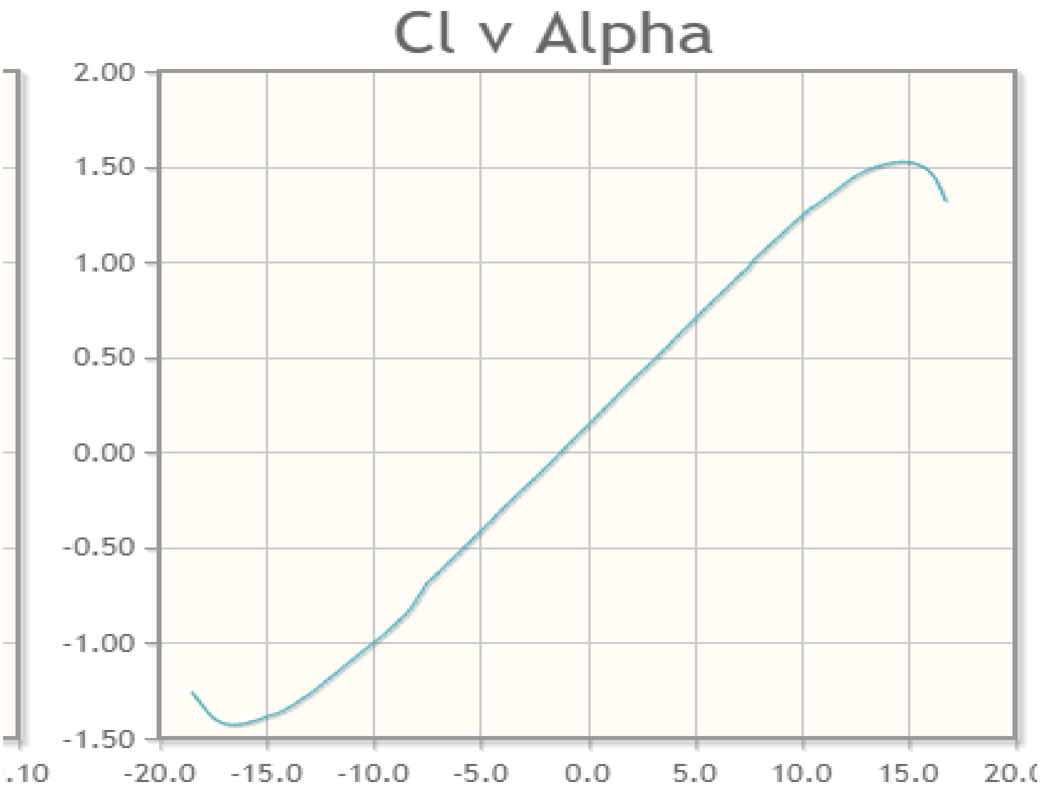
5.3 VALIDATION OF NACA 2412 AIRFOIL



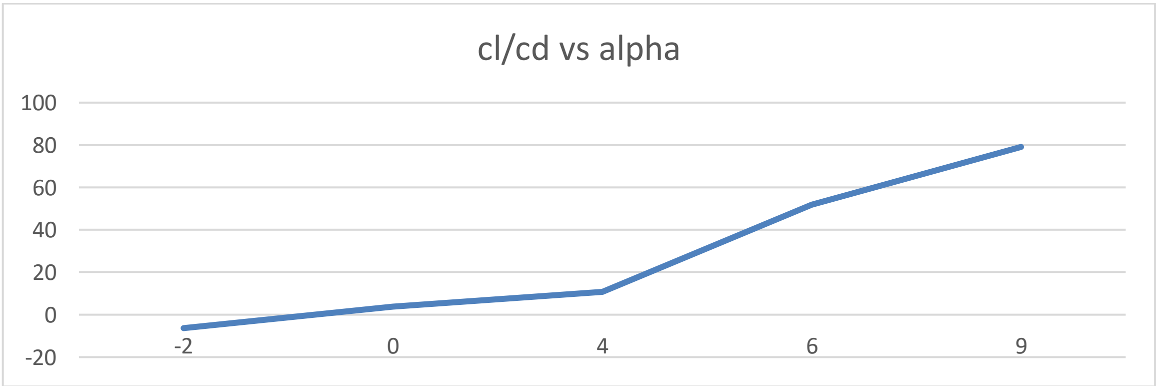
Graph 5.2 cl vs alpha NACA 2412 analysis results

- ➢ The Xfoil data shows a lift coefficient closely in the range of 0.7 at an angle of attack of 5 degrees.
- ➢ The analysis carried out mentions it to be the stall angle of attack with a lift coefficient around 0.6

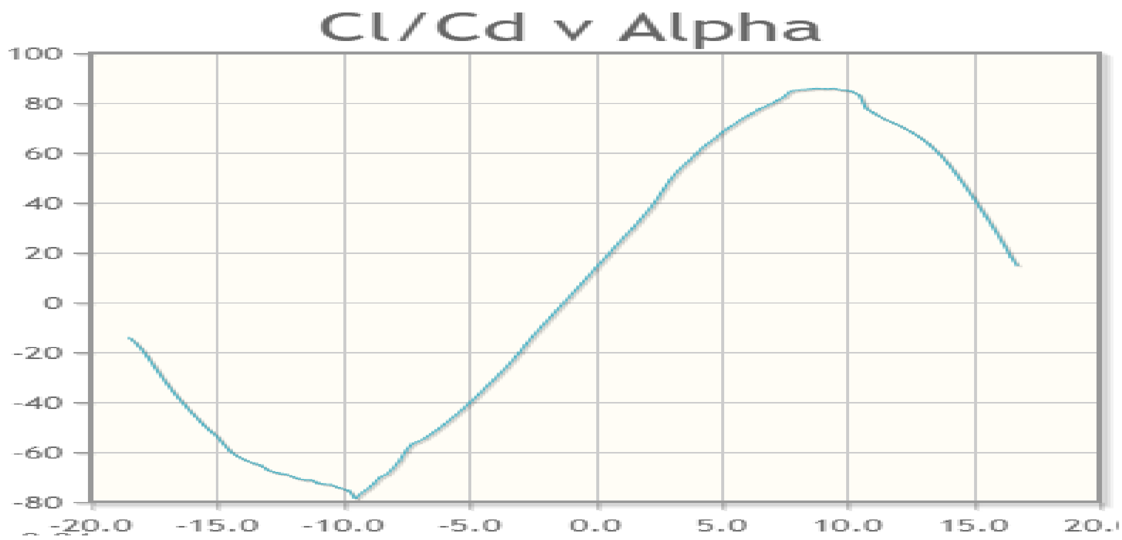
5.4 VALIDATION OF BOEING 737 WITH XFOIL DATA:



Graph 5.3 Cl vs alpha plot of Xfoil BOEING



Graph 5.4 Analysis result of cl/c vs alpha BOEING



Graph 5.5 Cl/ Cd vs alpha from Xfoil Data of BOEING

- The plot shows a lift coefficient of 1.5 at stall angle of attack at Mach almost negligible. However, with increase in Mach flow, the lift decreases and the lift coefficient obtained is near to 0.7125.
- The lift to drag ratio is maximum at 8 degrees of airfoil and is equal to 85.7
- The analysis showed a lift to drag ratio of 77 at 9-degree stall.

5.5 SUMMARY AND CONCLUSIONS:

The validations conclude that analysis has been satisfactorily employed and clearly demands more margin for accuracy. the results though initially showed deviations, working with the software over the course of time provided reliable results.

- It has been learnt that meshing or developing a wireframe in any analysis is as important as knowing the theory behind analysis.
- ANSYS provides a stable platform for finding the solution to a required convergence with different solution methods that can be employed depending on our requirement
- The airfoil BOEING exhibits a peculiar lift and drag parameters, and are employed in velocities nearing subsonic where compressibility is predominant
- BOEING airfoils are generally used. although with deviations from what we have analysed in commercial aircrafts.

- These exhibit good pressure gradient ranging some a negative pressure of 1000's to pressure of 10,000 Pascal.
- The stall angle of attack is found to be about 9 degrees which has been validated using Xfoil predictions
- It has been learnt that asymmetric airfoils often exhibit good lift because of their profile.
- Symmetric airfoils such as NACA 0012 has negligible lift and drag at 0-degree angle of attack
- It has been established that symmetric airfoils can only perform when the angle of attack is varied.
- NACA 0012 has a stall angle of 4 degrees when the flow is incompressible laminar.
- It can be inferred that NACA prescribed airfoils are a mere design experiments and are rather not employed in actual design since they perform erroneously in real time conditions of pressure, velocity and turbulence.
- NACA 2412 exhibits a high lift coefficient of 1.25 at 10 degrees but drag increase more predominantly.
- NACA 2412 is just confined to a minimal lift with real time conditions.

5.6 FUTURE SCOPE AND RESEARCH

With the advent of design software, more sophisticated ways of design are on a rise. It has made experimental analysis and mathematical analysis easier. However, generating the pseudocode for programming with UDF has been a challenge to graduate engineers.

More precise ways to analyse airfoil design can only be employed with thorough knowledge of design and theoretical analysis. When it comes to airfoil shapes, basic simulations that has been carried out is just a beginning to wider scope of aerodynamic analysis. Advanced mesh tools and advanced aerodynamic simulations are already in implementation. However, it depends on the accuracy with what it converges to the solution. 3D wing analysis, transient incompressible flow, high turbulence at hypersonic range of rockets are some of the advanced stages of research.

REFERENCES:

- http://airfoiltools.com/polar/details?polar=xf-b737a-il-1000000
- http://airfoiltools.com/airfoil/details?airfoil=naca0012h-sa
- http://prod.sandia.gov/techlib/access-control.cgi/1980/802114.pdf
- https://flugbox.wordpress.com/2015/04/
- https://aviation.stackexchange.com/questions/tagged/airfoil
- https://web.engr.uky.edu/~jbaker/ansystutor.html
- https://confluence.cornell.edu/display/SIMULATION/FLUENT+Learning+Module
- http://www.nptel.ac.in/courses/112104118/lecture-23/23-4_aerofoil.htm
- http://s6.aeromech.usyd.edu.au/aerodynamics/index.php/sample-page/subsonic-aerofoil-and-wing-theory/2-d-thin-aerofoil-theory/
- https://www.grc.nasa.gov/www/k-12/airplane/wrong1.html
- https://web.stanford.edu/~cantwell/AA210A_Course_Material/AA210A_Course_Notes/AA210_Fundamentals_of_Compressible_Flow_Ch_14_BJ_Cantwell.pdf
- http://164.100.133.129:81/econtent/Uploads/ACD2505-05-%20Airfoil%20Theory.pdf
- https://aerospace.illinois.edu/
- https://www.grc.nasa.gov/www/k-12/airplane/fuselage.html
- http://web.mit.edu/16.00/www/aec/flight.html
- https://www.nasa.gov/