Project Report

On

“Design and Analysis of Aerofoil Wing Structure for a Narrow Body Aircraft”

Submitted to

Brahmastra Aerospace Systems

Submitted by **Team Matar**

Shiva Sharma

Devarshi Amitabh Brahmakshatriya

Rithic C H

Dharmavendan

Under the guidance of

Mr. Roopesh Ravichandran

Event Coordinator, Design Engineer

Brahmastra Aerospace Systems

ACKNOWLEDGEMENT

We would like to express our special thanks of gratitude to our instructor and guide **Mr. Roopesh Ravichandran** and founder of Brahmastra Aerospace Systems **Mr. Subhash P Kuppusamy** who gave us the golden opportunity to do this project on the topic **"Design and analysis of Aerofoil and wing structure for a narrow body aircraft".** It helped us in doing a lot of research and we came to know about a lot of things related to this topic.

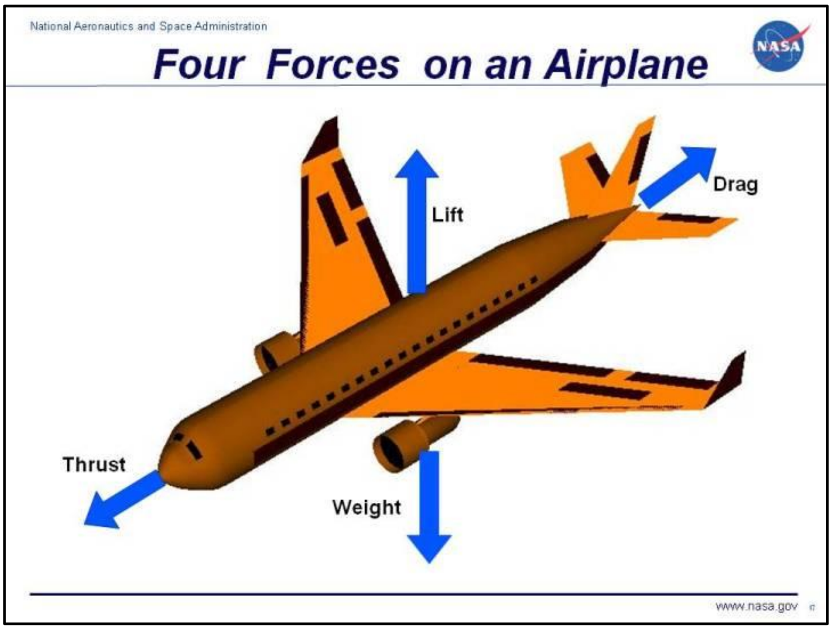
Finally, we would also like to thank our parents for constant motivation and group members for their continuous contribution and support during the project.

TABLE OF CONTENTS

Contents

INTRODUCTION

Aerodynamics is a study of how air moves around an object. It includes aircrafts, rockets and even automobiles. The four basic principles of aerodynamics are thrust, drag, lift, and weight. Of these, lift and drag are aerodynamic forces i.e., forces due to air over a solid body. Calculations of these quantities is often founded upon the assumption that the flow field behaves as a continuum, which in turn, is characterized by properties such as flow velocity, pressure, density, and temperature as functions of position and time.

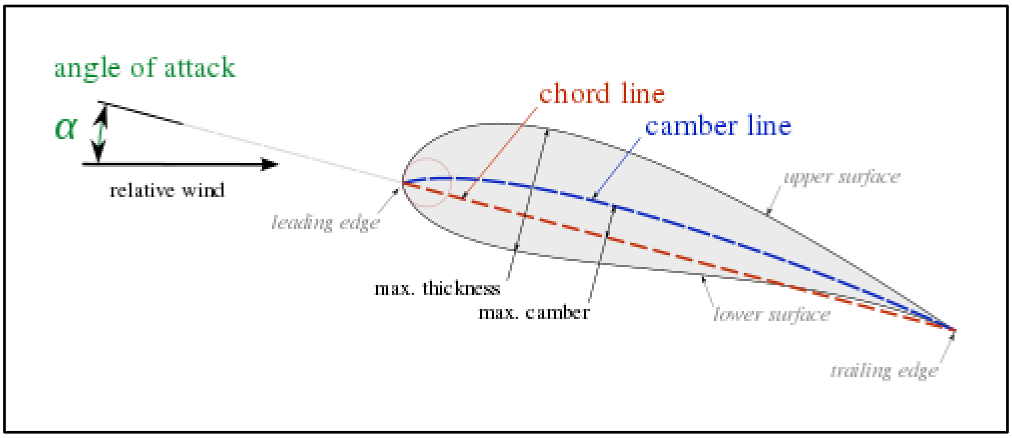


The aircraft is an engineering construction designed for flying over various distances in the atmosphere.

Airfoil is the cross-sectional shape of an object which is used to analyse its aerodynamic properties. There are two types of airfoil: Asymmetric and Symmetric Airfoils. Asymmetric airfoil has a flat bottom and a curved top. Symmetric Airfoil has identical upper and lower surfaces. The characterization of airfoil is specified below:-

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

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



Modern aerodynamics only dates to the seventeenth century, but aerodynamic forces have been harnessed by humans for thousands of years in sailboats and windmills. Fundamental concepts of continuum, drag, and pressure gradients appear in the work of Aristotle and Archimedes. In 1726, Sir Isaac Newton became the first person to develop a theory of air resistance, making him one of the first aerodynamicists. Dutch-Swiss mathematician Daniel Bernoulli followed in 1738 with hydrodynamics in which he described a fundamental relationship between pressure, density, and flow velocity for incompressible flow known today as Bernoulli's principle, which provides one method for calculating aerodynamic lift. In 1757, Leonhard Euler published the more general Euler equations which could be applied to both compressible and incompressible flows. The Euler equations were extended to incorporate the effects of viscosity in the first half of the 1800s, resulting in the Navier–Stokes equations. In 1799, Sir George Cayley became the first person to identify the four aerodynamic forces of flight, as well as the relationships between them, in 1871, Francis Herbert Wenham constructed the first wind tunnel, allowing precise measurements of aerodynamic forces. In 1889, Charles Renard, a French aeronautical engineer, became the first person to reasonably predict the power needed for sustained flight.

The aerodynamic of airfoils has been studied by Kutta (1902) on thin airfoils and Joukowski (1905) on airfoils with thickness. These airfoils were obtained from a circular cylinder by conformal mapping. The theory of Joukowski for 2D flow, so for an infinite wing, marks the beginning of the modern aerodynamic. At the same time, appeared the works of pioneers of the aviation, the Wright brothers, who based their study on the analysis of the stability of the plane made by Lanchester in 1894. They had a good understanding of wing and airfoil behaviour well beyond that of other experimenters at that time. Building on these developments as well as research carried out in their own wind tunnel, the Wright brothers flew the first powered airplane on December 17, 1903.

NACA was invented on March 13, 1915. NACA was established by the federal government through enabling legislation as an emergency measure during World War I to promote industry, academic, and government coordination on war-related projects. During the late 1920s and into the 1930s, the NACA developed a series of thoroughly tested airfoils and devised a numerical designation for each airfoil — a four-digit number that represented the airfoil section's critical geometric properties.

During the late 1920s and into the 1930s, the NACA developed a series of thoroughly tested airfoils and devised a numerical designation for each airfoil — a four-digit number that represented the airfoil section's critical geometric properties. By 1929, Langley had developed this system to the point where the numbering system was complemented by an airfoil cross-section, and the complete catalog of 78 airfoils appeared in the NACA's annual report for 1933. Engineers could quickly see the peculiarities of each airfoil shape, and the numerical designator ("NACA 2415," for instance) specified camber lines, maximum thickness, and special nose features. These figures and shapes transmitted the sort of information to engineers that allowed them to select specific airfoils for desired performance characteristics of specific aircraft.

Most of the success of the Wright brothers could be attributed to their own research, which used their wind tunnel and numerous experiments with controlled kites and gliders. In the same way, the experimental studies take a big progress in the wind tunnel thanks to tests on models made in England by Wenham and Philips.

A new step was crossed by Prandtl who created the theory of finite wing (1917-1918) that was presented with a laborious mathematical formula that honours nowadays the domain of aerodynamics. This theory gave a new and a big progress to aerodynamics. For twenty years, the theories of Joukowski and Prandtl have been the starting points of many theoretical and experimental works and permit us in this context to describe the aerodynamic properties of the elliptical wing and of arbitrary plan form. This elliptical wing leads to an also elliptical distribution for the local circulation and to the famous minimum induce drag . One can represent any arbitrary distribution for local circulation by means of the Fourier series development. When the distribution of local circulation is not elliptical, the induce drag is not minimal but it appears that the difference is relatively small.

The report discusses on the design and analysis of NACA 0012 symmetrical airfoil which is best suited for the narrow body aircraft.

LITERATURE REVIEW

[1] R.K. Ganesh Ram, Yashaan Nari Cooper, Vishank Bhatia, R. Karthikeyan, C. Periasamy, “Design Optimization and Analysis of NACA 0012 Airfoil Using Computational Fluid Dynamics and Genetic Algorithm”.

[2] McCroskey, W. J.[1987], “A Critical Assessment of Wind Tunnel Results for the NACA 0012 Airfoil”

[3] Patel Karna S., Patel Saumil B., Patel Utsav B., Prof. Ahuja Ankit P. [2017] , “CFD Analysis of an Airfoil”.

[4] Jini Raj [2017], “CFD analysis of flow characteristics of NACA0012 airfoil using SU2”.

[5] Critzos, Chris C. Heyson, Harry H. Boswinkle, Robert W., jr [1955], “Aerodynamic Characteristics of NACA 0012 Airfoil Section at Angles of Attack From 0 deg to 18 deg”.

[6] Talluri Srinivas Rao, Trilochan Mahapatra, Sri Chaitanya Mangavelli [2018], “Enhancement of Lift-Drag characteristics of NACA 0012”.

[7] [9] Da Xiao, “Numerical Simulation of Flow Past NACA 0012 Airfoil Using a Co-Flow Jet at Different Injection Angles to Control Lift and Drag”, Washington University in St. Louis.

[8] Charles D. Harris [1981], “Two-Dimensional aerodynamic characteristics of the NACA 0012 airfoil in the Langley 8-foot transonic pressure tunnel”.

DESIGN AND ANALYSIS OF NACA 0012 AIRFOIL

NACA Airfoil

NACA is the abbreviation of **National Advisory Committee for Aeronautics** , a predecessor to NASA. It was founded on March 3, 1915, to undertake, promote, and institutionalize aeronautical research.

During World War II, NACA was described as "The Force Behind Our Air Supremacy" due to its key role in producing working superchargers for high altitude bombers, and for producing the laminar wing profiles for the North American P-51 Mustang.

NACA was also key in developing the area rule that is used on all modern supersonic aircraft and conducted the key compressibility research that enabled the Bell X-1 to break the sound barrier. Symmetrical airfoil is designed and developed by the 'National Advisory Committee for Aeronautics'.

NACA airfoils are available in four-digit, five-digit, six-digit and seven-digit series.

Four- and five-digit series airfoils can be modified with a two-digit code preceded by a hyphen in the following sequence:

- One digit describing the roundness of the leading edge, with 0 being sharp, 6 being the same as the original airfoil, and larger values indicating a more rounded leading edge.
- One digit describing the distance of maximum thickness from the leading edge in tenths of the chord.

For example, the NACA 1234-05 is a NACA 1234 airfoil with a sharp leading edge and maximum thickness 50% of the chord (0.5 chords) from the leading edge. In addition, for a more precise description of the airfoil all numbers can be presented as decimals.

NACA Nomenclature

The NACA 0012 airfoil is the symmetrical airfoil designed and developed by the **'National Advisory Committee for Aeronautics'**. The airfoil are generated by using a series of number like 4-digit airfoil, 5-digit airfoil, 6-digit airfoil and so on.

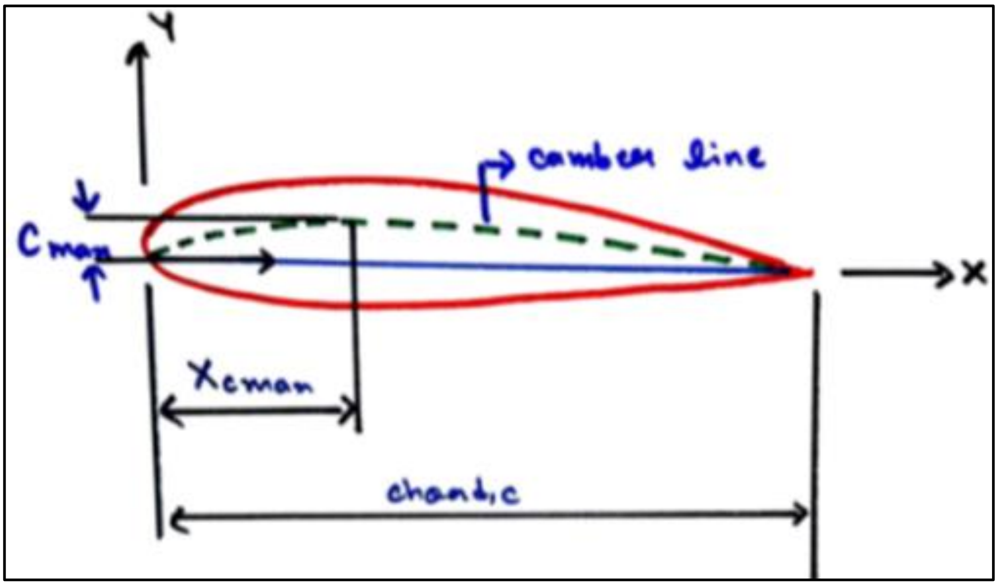
The NACA 0012 symmetrical airfoil consist of the following 4 digits: -

0: The first digit represents the maximum camber thickness (C_{max}) in the terms of percentage of the total chord.

0: The second digit represents the maximum camber position for the leading edge in terms of tenths of the chord length. (X_{cmax})

12: The last two digits represents the 't/c' ratio i.e., thickness to chord ratio of the airfoil.

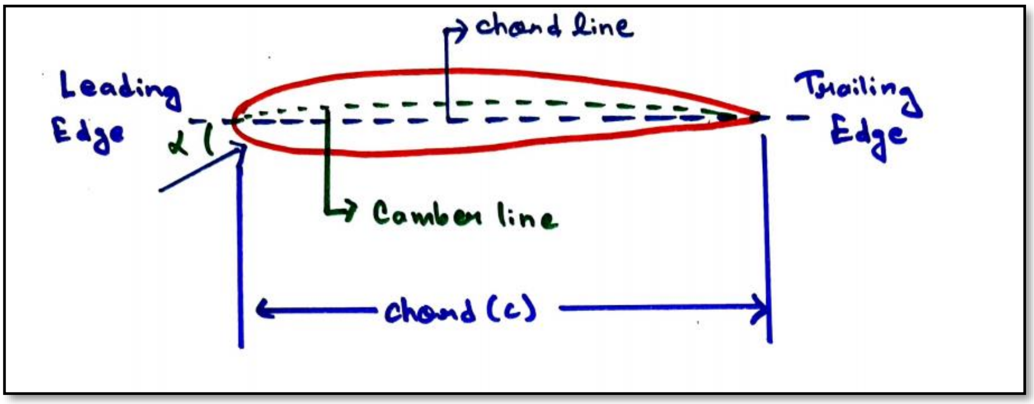
The flow field pattern for the symmetrical airfoil is much simpler if compared with the cambered airfoil and the supercritical airfoil. The symmetrical airfoil has **zero camber** and within the framework of the **Thin Airfoil Theory,** the symmetrical airfoil is treated as a flat plate.



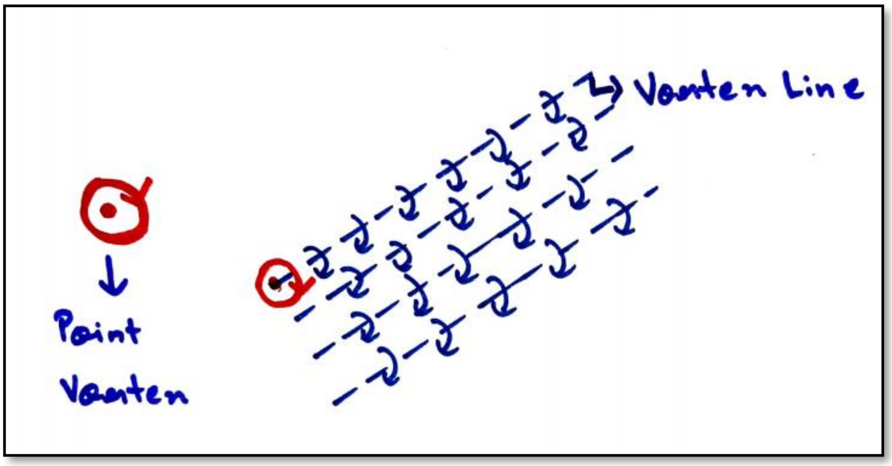
Glauert's Thin Airfoil Theory

The basic assumptions made in the thin airfoil theory are: -

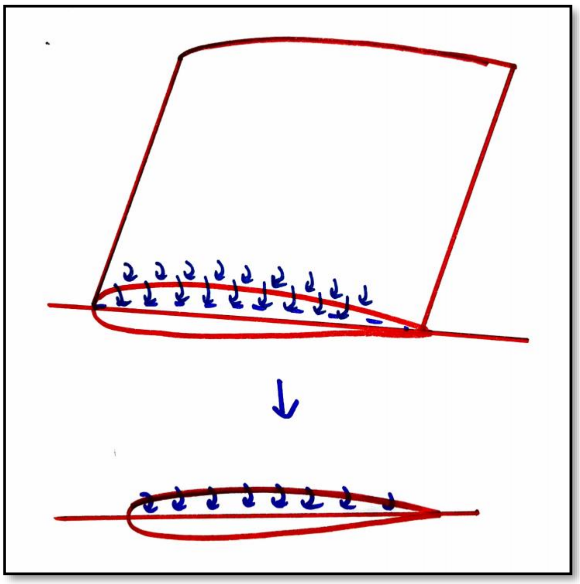
1. The thickness of the airfoil is exceedingly small i.e., the camber line is very close to the chord line.
2. The angle of attack is also very small.



Basically, in the thin airfoil theory, the upper section of the airfoil needs to be replaced with the vortex sheet. This is possible by combining the vortex lines which results in the formation of the roller sheets and the surface vortex gets generated.



The camber line is then replaced by the vortex line and will have the significant effect of the upper and the lower surface.



The derivation of the **Fundamental Lifting Line Theory** is discussed below.

In the Prandtl's classical lifting line theory, a vortex filament of strength Γ is bound onto a fixed position in the flow field. This will be known as the bound vortex and from the **Kutta-Joukowski Theorem,** a lift force would be thereby generated $L=\rho_\infty.V_\infty.\Gamma_\infty$.

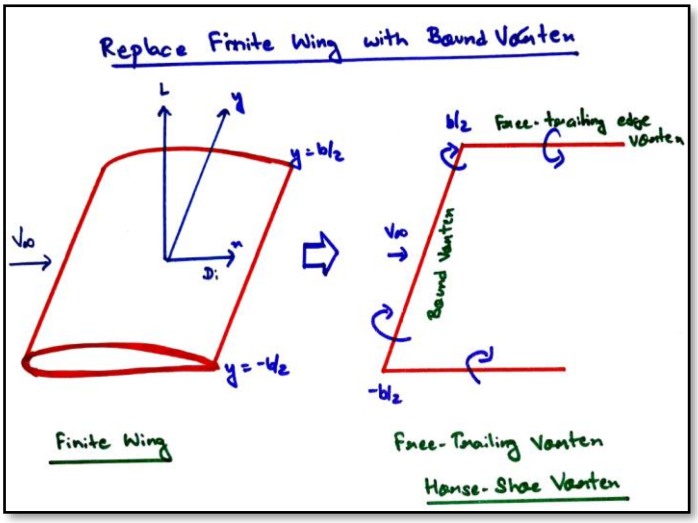


Figure 1. Finite Wing Effect Replaced by Horseshoe Vortex

The bound vortex generated will move with the fluid (air) elements throughout the entire flow.

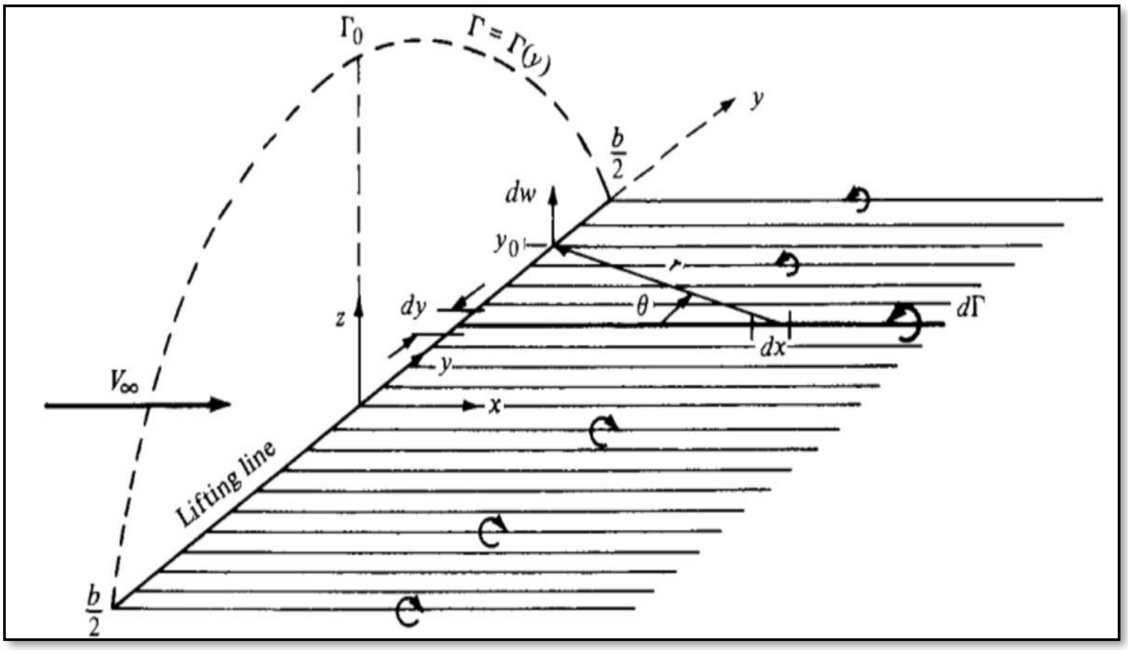
The finite wing is then replaced with the bound vortex extending from y=-b/2 to y=+b/2.

The downwash velocity W(y) is given by,

$$w(y) = -\frac{\Gamma}{4\pi(b/2+y)} - \frac{\Gamma}{4\pi(b/2-y)}$$

$$w(y) = -\frac{\Gamma}{4\pi}\frac{b}{(b/2)^2 - y^2}$$

Now we consider the superposition of infinite number if the horse-shoe vortices attached with the lifting line.



Total Induced Velocity or $Wy_{(o)}$, is given by,

$$w(y_0) = -\frac{1}{4\pi}\int_{-b/2}^{b/2}\frac{(d\Gamma/dy)\,dy}{y_0 - y}$$

$$\alpha_i(y_0) = \frac{1}{4\pi V_\infty}\int_{-b/2}^{b/2}\frac{(d\Gamma/dy)\,dy}{y_0 - y}$$

Above is an expression for the induced angle of attack in terms of the circulation distribution Γ(y) along the wing.

Now, $\frac{2\Gamma(y_0)}{V_\infty . c} = 2\pi \left(\alpha - \alpha_i(y_0) - \alpha_{L=0} \right)$

$$\alpha(y_0) = \frac{\Gamma(y_0)}{\pi V_\infty c} + \alpha_i(y_0) + \alpha_{L=0}$$

Hence,

$$\boxed{\underbrace{\alpha(y_0)}_{\alpha_{geometric}} = \underbrace{\frac{\Gamma(y_0)}{\pi V_\infty . c(y_0)} + \alpha_{L=0}(y_0)}_{\alpha_{effective}} + \underbrace{\left(\frac{1}{4\pi V_\infty} \int_{-b/2}^{+b/2} \frac{(d\Gamma/dy).dy}{(y_0 - y)} \right)}_{\alpha_{induced}}}$$

Hence, it can be finally concluded from the **Prandtl's Classical Lifting Line Theory** that '**The geometric angle of attack is equal to the sum of effective angle of attack and the induced angle of attack'.**

Computational Fluid Dynamics (CFD)

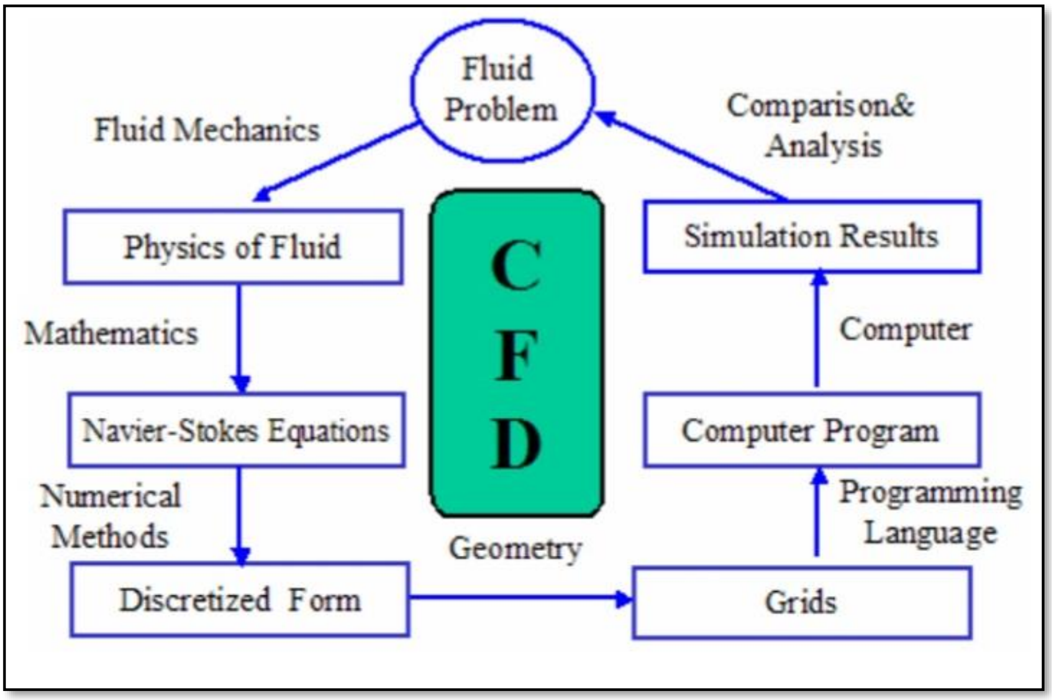
Computational Fluid Dynamics is the analysis of the engineering systems involving the fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation i.e. ANSYS.

In other words, Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that analyses and solves problems regarding the flow of fluid with the help of numerical analysis and data analysis. It is a branch of CAE that can simulate motions of fluids and heat transfer using numerical approaches. Calculations are performed with the help of computers which are required in order to simulate the free stream flow of the fluid, interaction of the fluids with surfaces being defined by boundary conditions.

Thanks to the advent of supercomputers, better results can be achieved now which helps massively in solving large and complicated problems. Thanks to the advancement of technology especially in software, the accuracy and speed of solving complicated situations like transonic and turbulent have remarkably improved. This however does not mean CFD completely replaces other means of testing.

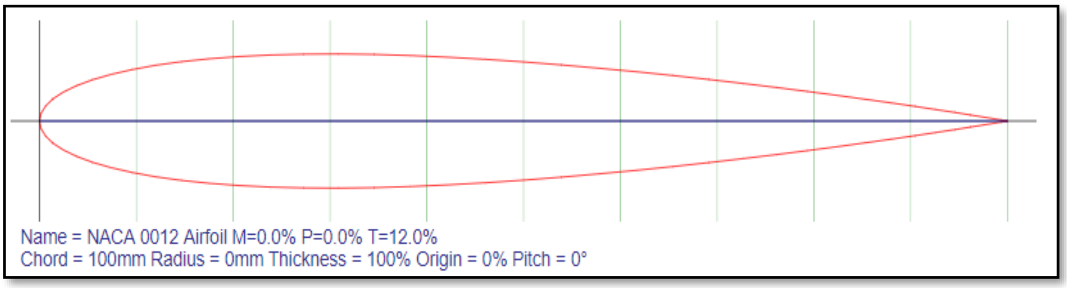
Applications in CFD

CFD is useful in a wide variety of applications and here we note a few to give you an idea of its use in industry. The simulations shown below have been performed using the **FLUENT** software. CFD can be used to simulate the flow over a vehicle. Broadly, the strategy of CFD is to replace the continuous problem domain with a discrete domain using a grid. In the continuous domain, each flow variable is defined at every point in the domain. In the discrete domain, each flow variable is defined only at the grid points. In a CFD solution, one would directly solve for the relevant flow variables only at the grid points. The values at other locations are determined by interpolating the values at the grid points. The governing partial differential equations and boundary conditions are defined in terms of the continuous variables ρ, V etc. One can approximate these in the discrete domain in terms of the discrete variables. The discrete system is a large set of coupled, algebraic equations in the discrete variables. Setting up the discrete system and solving it (which is a matrix inversion problem) involves a very large number of repetitive calculations, a task we humans palm over to the digital computer.



Design of NACA 0012 Airfoil

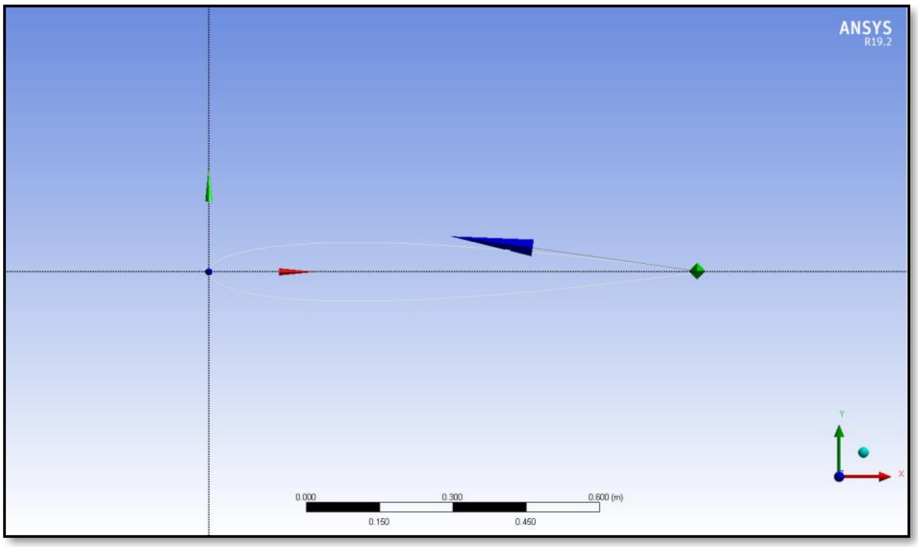
The coordinates of the typical NACA 0012 airfoil are plotted with 100 points, 51 points on the upper surface and 50 on the lower surface. The following configuration was analysed: -



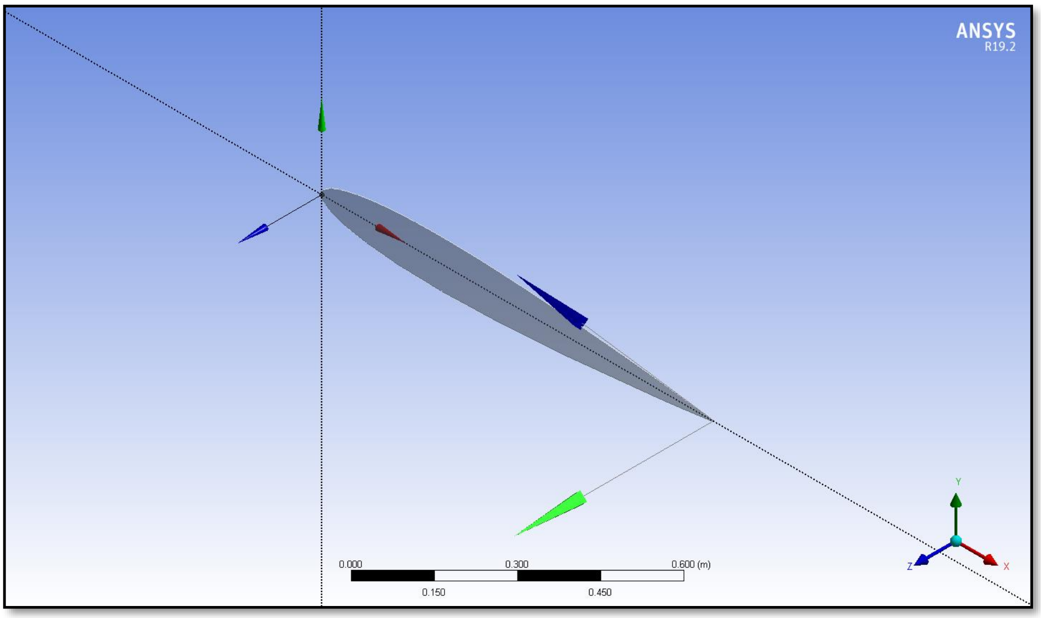
The coordinated for the NACA 0012 airfoils are imported from ‘NACA Airfoils Tool Plotter’.

The ANSYS Workbench R19.2 version is used for initial creation of the geometry. ANSYS is an American Public Company and was founded in 1970. It is basically a Computer **Aided Engineering Software (CAE)** and is used by various companies to study the real time flow filed analysis over the bodies, determination of aerodynamic forces like lift, drag, etc. It also helps to simulate the flow over the airfoils in a used defined pattern to observe the flow filed characteristics and the stress concentration region.

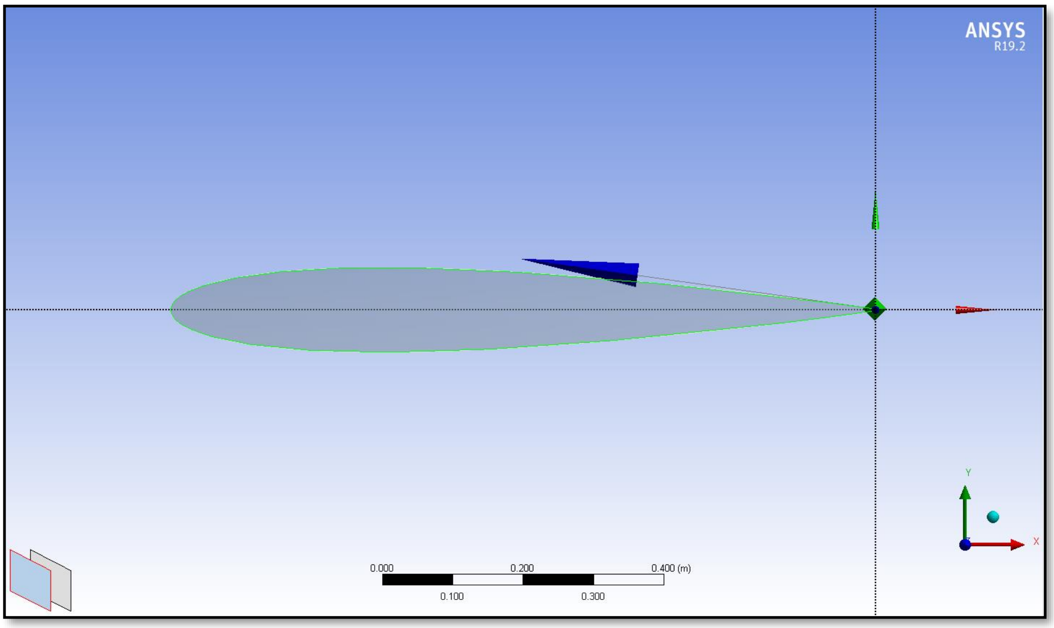
The axis of design should be selected properly to get better viewing angle, which allows various operations of sketching on the 2D sketch.



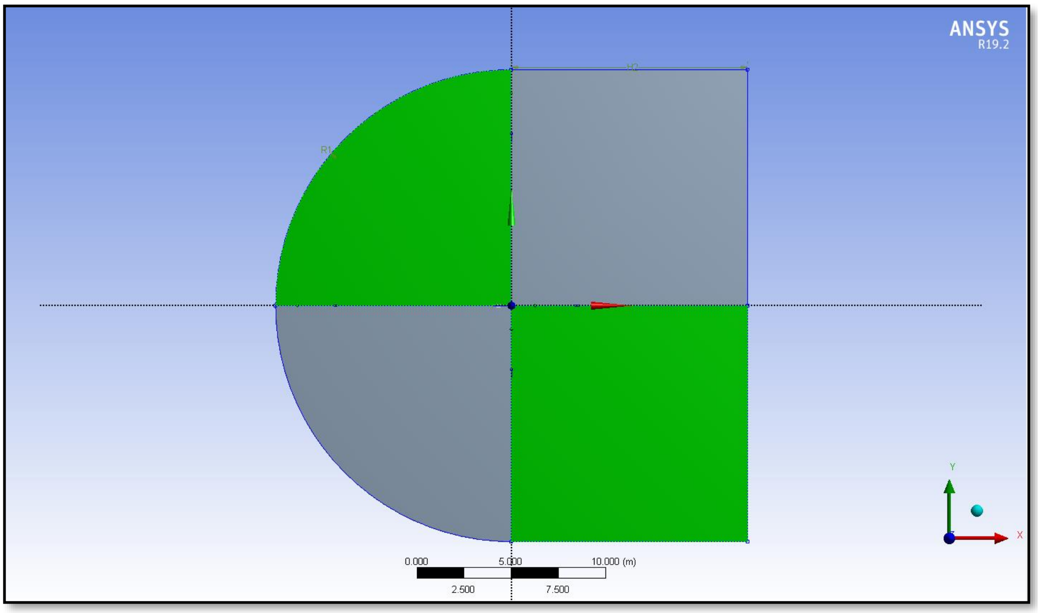
The edges of the airfoil are closed in order to properly analyse the flow field.



The origin axis which located at the leading edge of the airfoil, needs to be shifted near the trailing edge location for proper application of boundary condition.



C-Domain is required around the airfoil to visualize the flow field around the airfoil. The C-Domain will act as the boundary in which the ANSYS Fluent will run the calculations and provide the analysis results



The NACA 0012 airfoil needs to be divided into exceptionally fine elements at the leading edge and the trailing edge to get very accurate results during the simulation. High quality meshing requires more CPU time but gives accurate results up to 99.99%.

Next stage is the meshing of the airfoil and the domain. Meshing is the process in which the continuous geometric space of an object is broken down into thousands or more of shapes to properly define the physical shape of the object. The more detailed a mesh is, the more accurate the 3D CAD model will be, allowing for high fidelity simulations. Meshing, also known as mesh generation, is the process of generating a two-dimensional and three-dimensional grid; it is dividing complex geometries into elements that can be used to discretize a domain. Since meshing typically consumes a significant portion of the time in acquiring simulation results, advanced automated meshing tools can provide faster and more accurate solutions. Creating a high-quality mesh is one of the most critical factors that should be considered to ensure simulation accuracy. Creating the most appropriate mesh is the foundation of engineering simulations because the mesh influences the accuracy, convergence, and speed of the simulation. Computers cannot solve simulations on the CAD model's actual geometry shape as the governing equations cannot be applied to an arbitrary shape.

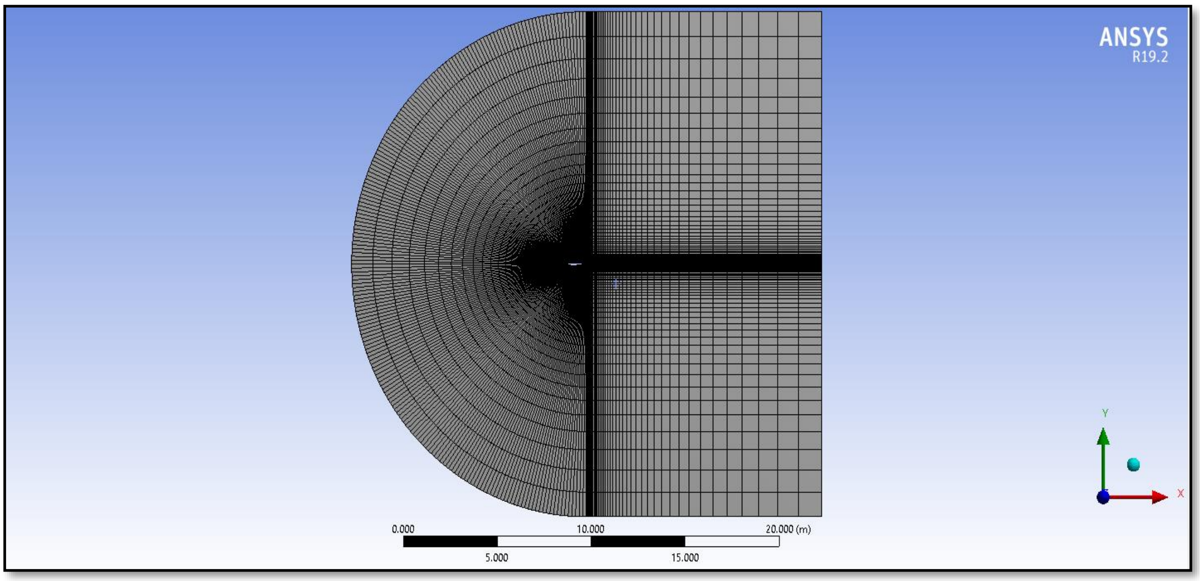


Figure 2. Final Meshing of the Domain

The above figure shows that the meshing of the domain is very fine at the LE, TE, pressure surface and the suction surface. It will provide better plots and accurate results while performing computational fluid dynamics.

Once, the meshing is generated, the **Inlet, Outlet and Airfoil** is named using the **named selection** option. This was the final stage in the meshing tab and the domain is then ready for the analysis in the CFD simulator i.e. **FLUENT**.

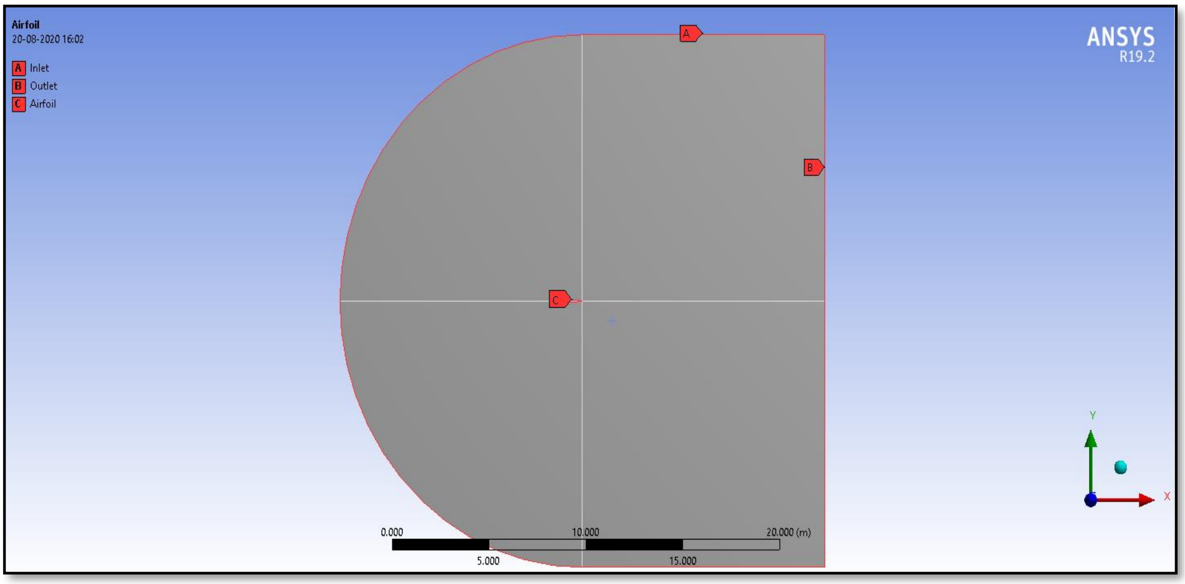
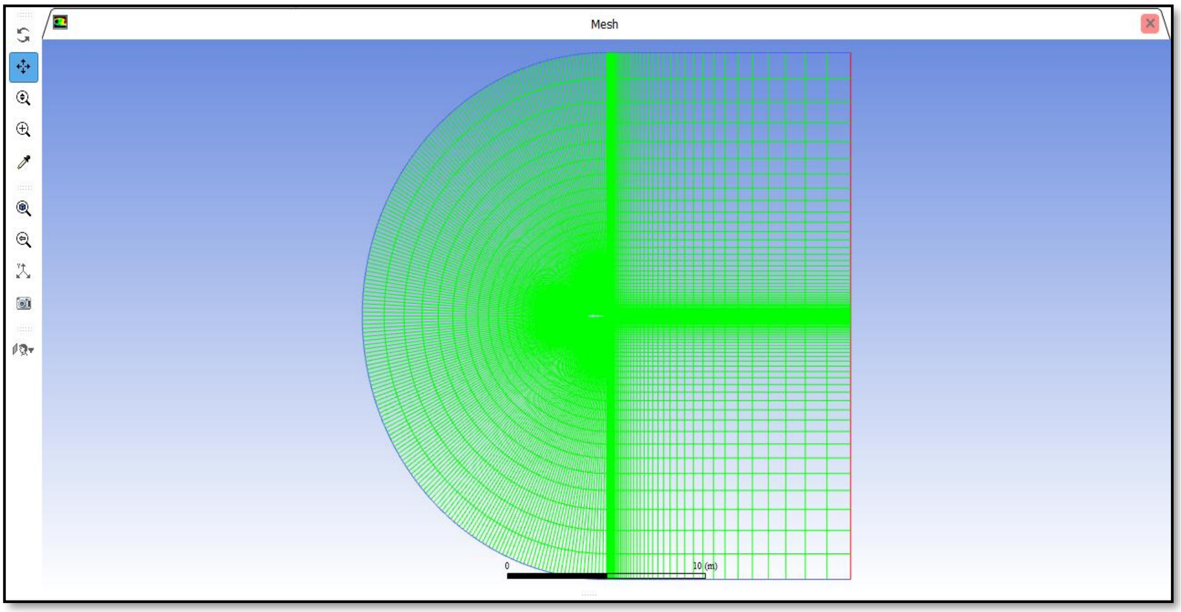


Figure 3. Named Selection of the Domain

Analysis of NACA 0012 using ANSYS Fluent

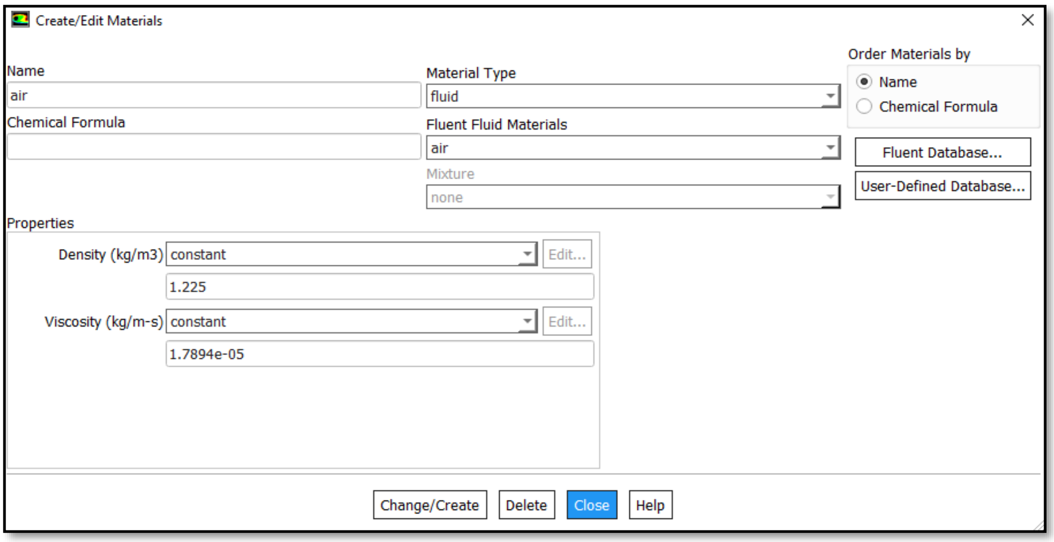
After the completion of the meshing stage, the model or the domain is then transferred to the solver. The solver settings must be precisely selected so that the model is compatible and produce precise results with the given input parameters.



From the above figure, it is clear that the airfoil is acting as a wall and the flow will pass over the NACA-0012 airfoil. The effect of the flow will only be considered in the domain consisting of a semi-circular arc and a closing rectangle at the end.

The Spalart Allmaras is one of the best known models for solving the flow over 2D airfoils. This model specifies the turbulent flow to be calculated for a given specific airfoil. This model will be used to generate the results on airfoil.

The next stage step is to specify the medium of the flow while performing analysis.. The medium selected for the analysis of NACA-0012 airfoil is **air,** with a stand value of density as **1.225 kg/m^3.**

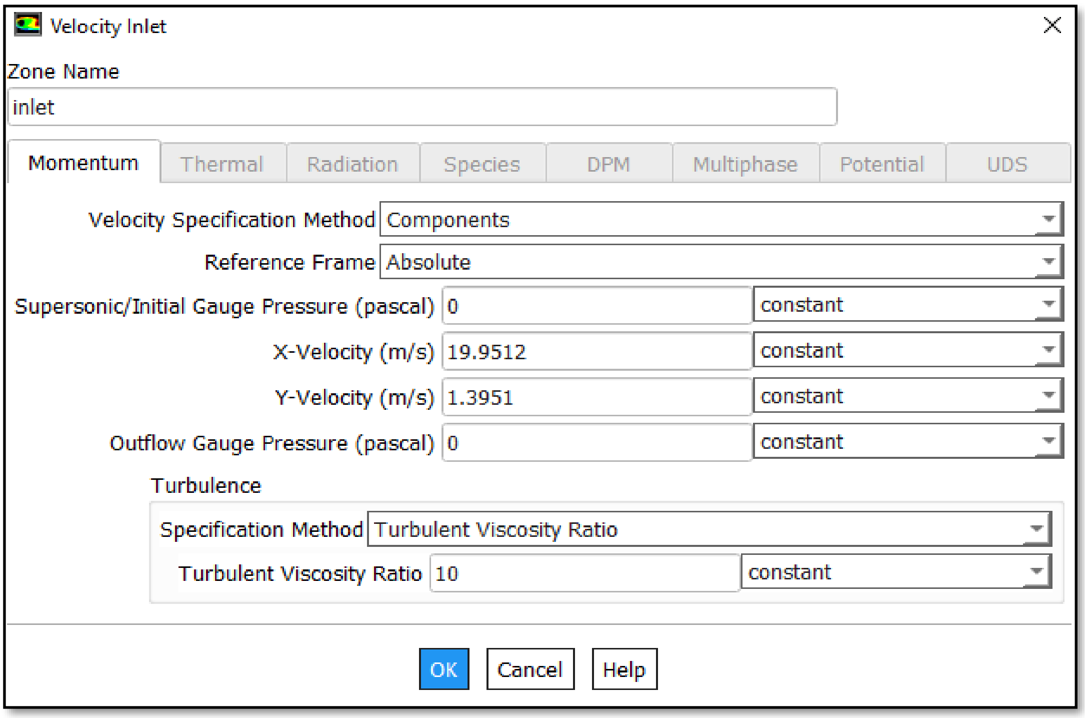


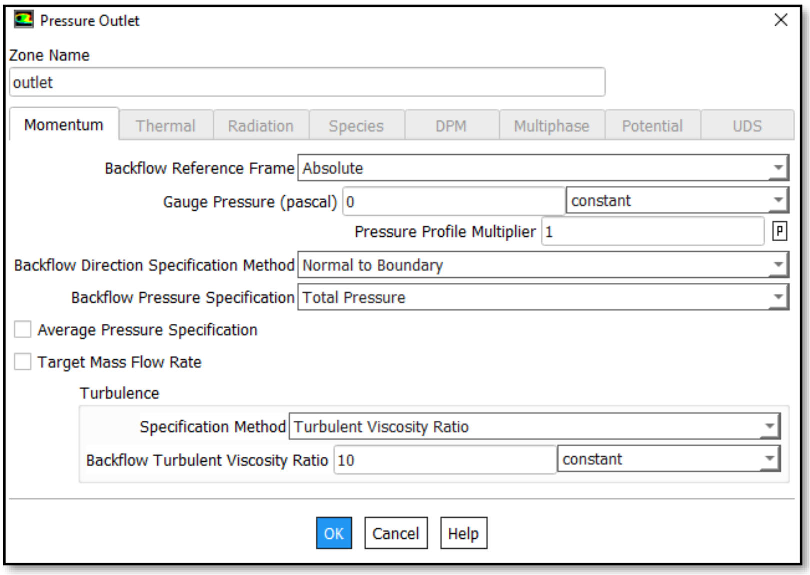
The analysis will be done at a flight velocity of **20 m/s** with angle of attack at **6º**.

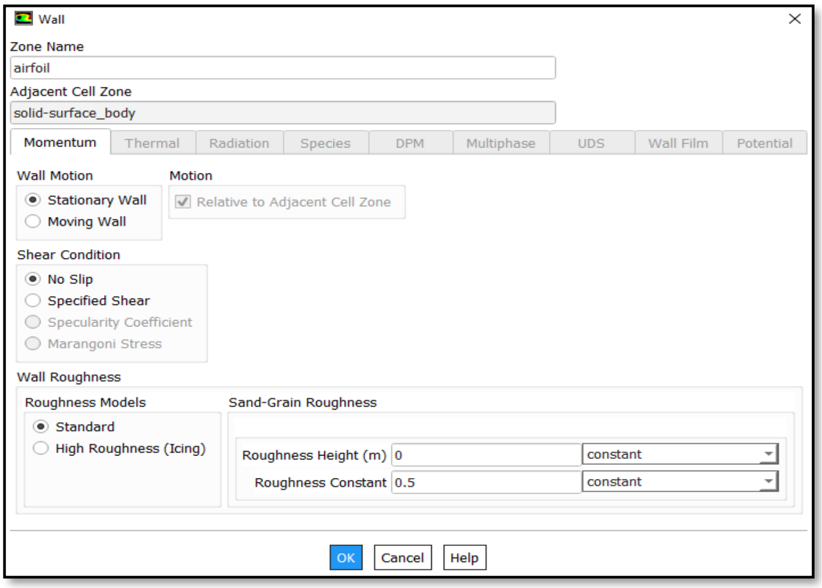
Boundary conditions for the selected airfoil and the domain are specified below:-

The boundary conditions are basically applied at 3 sections :

- Inlet of C-Domain
- Outlet of C-Domain
- Airfoil (Region of Flow Field Analysis)



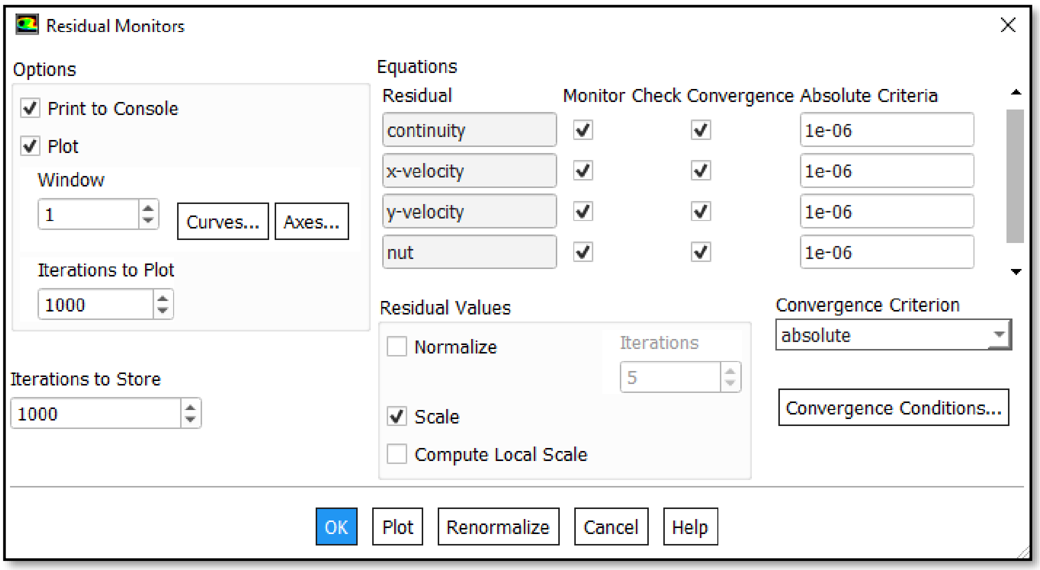




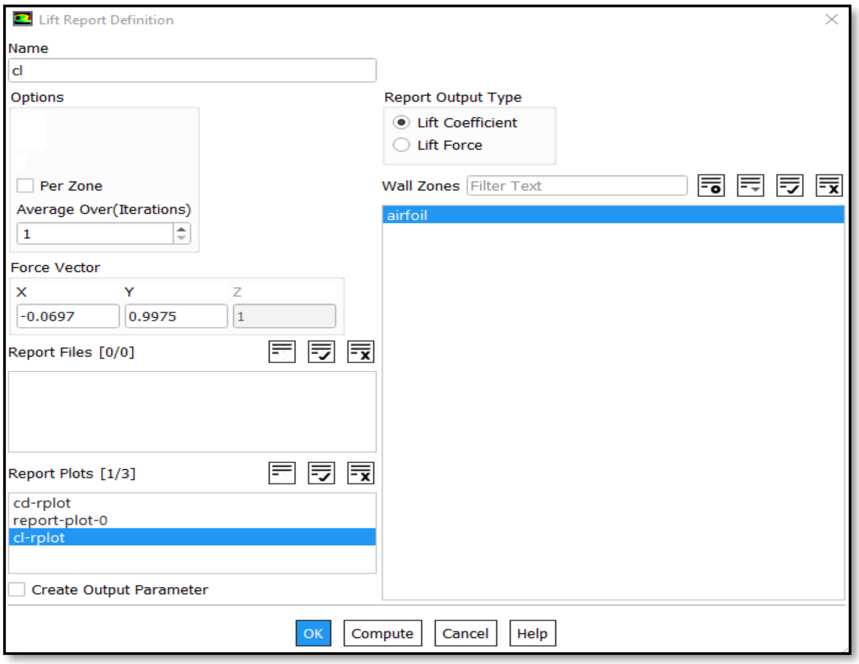
The next stage in the analysis part is to specify the reference position and the values. It is essential to specify inlet as all the computation work start from the boundary conditions specified at the inlet. While selecting the inlet as the initial parameters, the reference will be automatically added as specified by the user in the earlier stages.

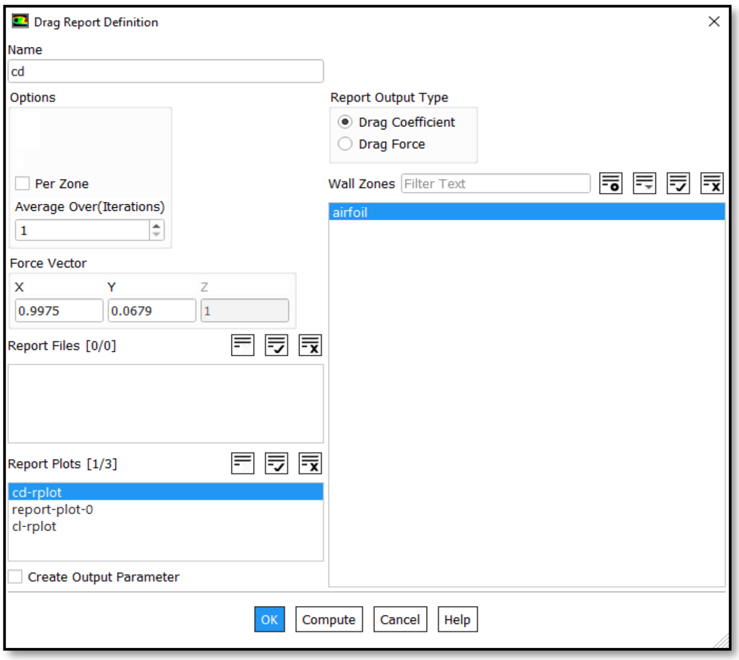
For obtaining good results, the residuals must converge smoothly over the specified number of iterations provided.

For the solution, the residuals are set at 10e-6 for smooth convergence.

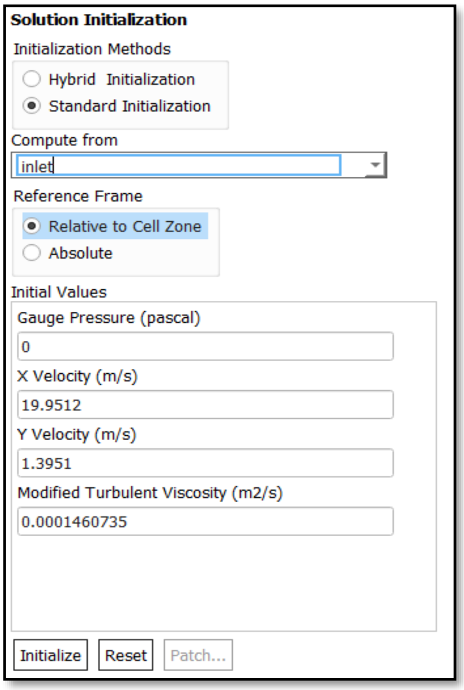


The main aspect is to determine the coefficient of lift (cl) and ‘coefficient of drag (cd). For obtaining the results of coefficient of lift and drag, it is required to substitute the values of the force vectors. The force vectors are divided into two directions. X-Force vector and the Y-Force vector. The cosine and the sine components involved in the force components should be carefully placed with the signs to obtain accurate values of the aerodynamic coefficients.



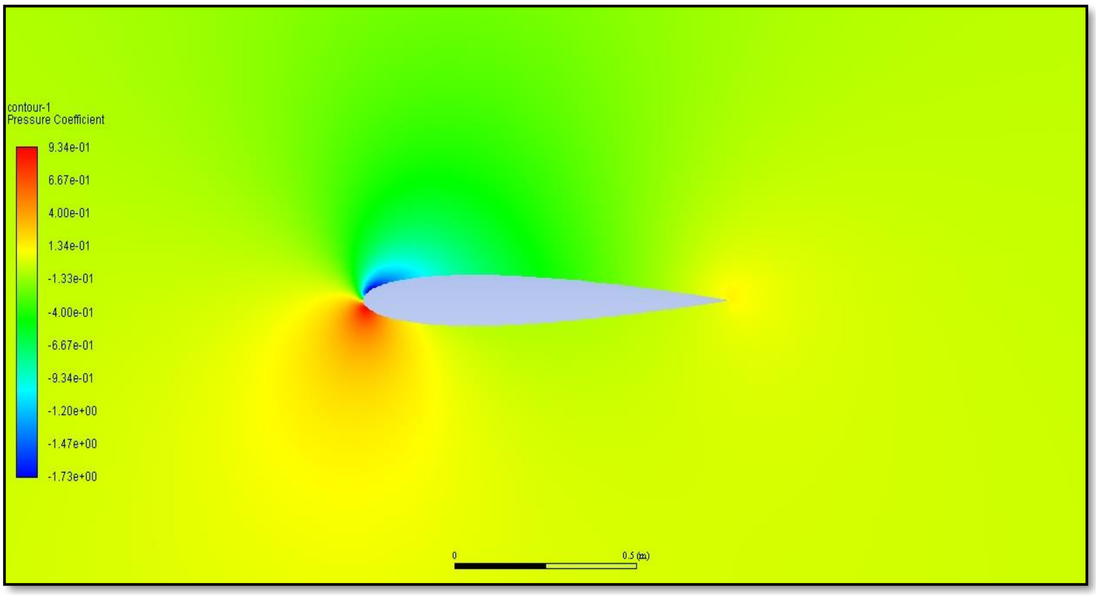


Before proceeding with the initialization of the solution, the setup must be calculated for the inlet region as the velocity is travelling from inlet to outlet of the C-Domain.

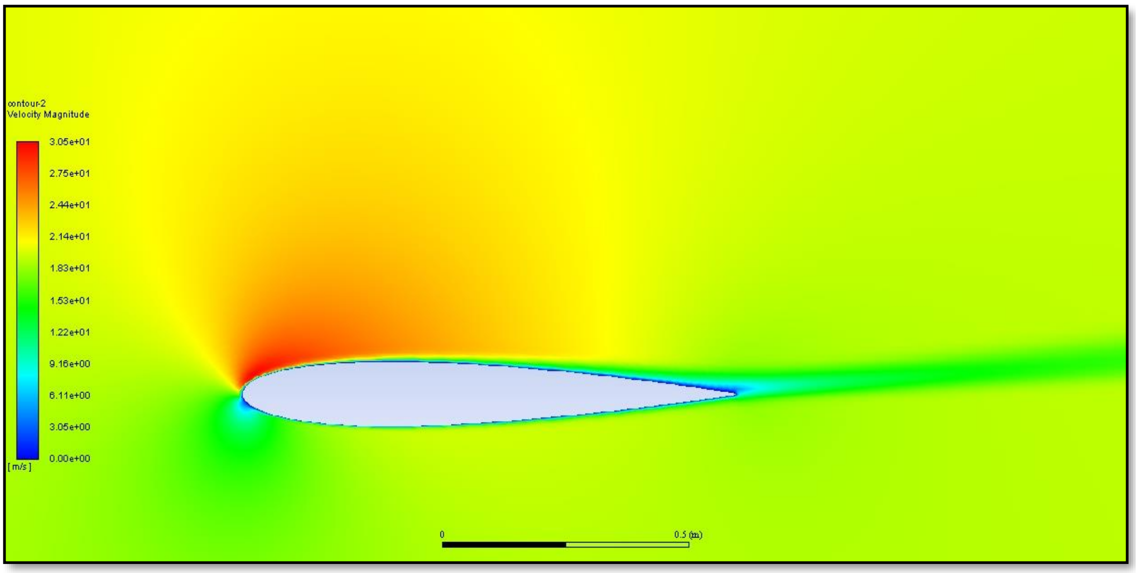


The solution is then initialized, and calculation is done for the determination of lift coefficient, drag coefficient and pressure distribution over NACA 0012 airfoil.

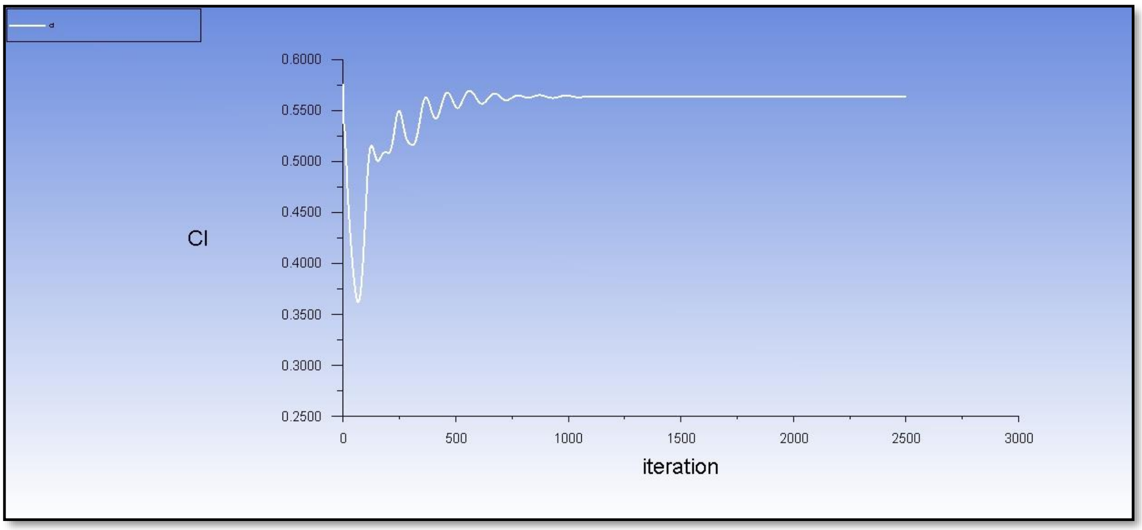
The pressure coefficient contour of NACA 0012 airfoil,



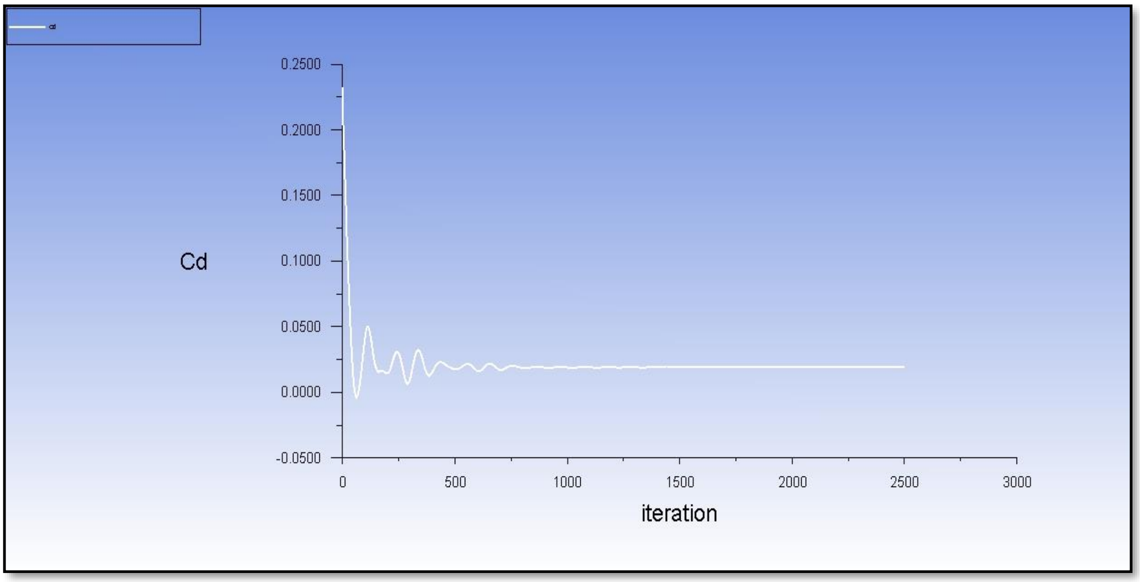
The velocity magnitude contour of NACA 0012 airfoil,



Lift Coefficient plot for NACA-0012 airfoil at 6^{o} angle of attack,



Drag Coefficient plot for NACA-0012 airfoil at 6^{o} angle of attack,



Analysis Results of NACA 0012 Airfoil

The analysis was performed at inlet velocity of 20 m/s and angle of attack of 6 degree, the following results were obtained: -

1. Coefficient of Lift (Cl): 0.56
2. Coefficient of Drag (Cd): 0.019
3. Highest Velocity Magnitude: 30.5 m/s
4. Value of Pressure Coefficient on Lower Surface (Cp)= 0.93
5. Value of Pressure Coefficient on Upper Surface (Cp)= -1.73

The plots were generated for pressure coefficient, velocity magnitude, velocity vectors, plots of coefficient of lifts and drags and the convergence criteria plot of the scaled residuals.

The results obtained at 6-degree angle of attack with inlet velocity of 20 m/s shows that the NACA 0012 airfoil has a symmetrical lift distribution with a maximum lift coefficient of 1. **Hence, the angle of attack increases with the increase in angle of attack before it reaches to its critical value where stall occurs.**

REFERENCES

[1] Da Xiao, “Numerical Simulation of Flow Past NACA 0012 Airfoil Using a Co-Flow Jet at Different Injection Angles to Control Lift and Drag”, Washington University in St. Louis.

[2] A. Merabet and B. Necib “Characterisation of Wings with NACA 0012 Airfoils”, Mechanical Engineering Department, Laboratory of Mechanics, Mentouri University – Constantine 25000 – Algeria.

[3] Nipun P Raval, Mehta Malay, Lalani Jitesh, “CFD Analysis of NACA0012 Aerofoil and Evaluation of Stall Condition”, Institute of Technology Nirma University, S-G Highway, Ahmedabad Gujarat.

[4] Vishank Bhatia[1], Ramanujam Karthikeyan[2], Periasamy Chinnapalaniandi[3], “Design Optimization and Analysis of NACA 0012 Airfoil Using Computational Fluid Dynamics and Genetic Algorithm”, [1]University of California, [2]Los Angeles, BITS Pilani, Dubai, [3]BITS Pilani, Dubai

[5] Nipun P Raval, Mehta Malay, Lalani Jitesh, “CFD Analysis of NACA0012 Aerofoil and Evaluation of Stall Condition”, Institute of Technology Nirma University, S-G Highway, Ahmedabad Gujarat.

[6] Novel Kumar Sahu, Mr.Shadab Imam, “Analysis of Transonic Flow over an Airfoil NACA0012 using CFD”, Department of Mechanical Engineering, Christen College of Engineering & Technology, Bhilai, CG.

[7] “ANSYS Fluent Tutorial Guide”, ANSYS, Inc. Release 15.0, Southpointe 275 Technology Drive.