CFD FLOW ANSLYSIS ON NACA 23012, NACA 23014 AND NACA 63415 AEROFOILS USING ANSYS FLUENT

An Internal Project report submitted in partial fulfillment of the requirement
For the award of the degree of

BACHELOR OF TECHNOLOGY
IN
MECHANICAL ENGINEERING

Submitted by

N.SUDHIN RAJ	**(318126520195)**
P.SRINIVAS	**(318126520197)**
Y.MURALI	**(319126520L38)**
P.GANESH	**(319126520L37)**
M.AKHIL	**(319126520L50)**

Under the Esteemed Guidance of

Mr. J.V BHANUTEJ, M.Tech , (Ph.D.)
Assistant Professor

DEPARTMENT OF MECHANICAL ENGINEERING
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES (A)
(Affiliated to Andhra University, Approved by AICTE, Accredited by NBA, NAAC
SANGIVALASA, VISAKHAPATNAM (District) – 531162

2022

ANIL NEERUKONDA INSTITUTE OF TECHNOGY & SCIENCES &TECHNOLOGY

(UGC Autonomous & Affiliated to Andhra University)

Sangivalasa, Bheemunipatnam (M), Visakhapatnam (Dt)

PRAGNANAM BRAHMA
ANITS

CERTIFICATE

This is to certify that the Project Report entitle **"CFD FLOW ANSLYSIS ON NACA 23012,NACA 23014 AND NACA 63415 AEROFOILS USING ANSYS FLUENT "** has been carried out by **N.SUDHIN RAJ (318126520195), P.SRINIVAS (318126520197), Y.MURALI (319126520L38), P.GANESH (319126520L37), M.AKHIL (319126520L50)** under the esteemed guidance of Prof **J.V.BHANU TEJ**, in partial fulfillment of the requirements for the award of the Degree of Bachelor of Technology in Mechanical Engineering by Andhra University, Visakhapatnam.

APPROVED BY

28.5.22

Dr. B. NAGA RAJU
Head of the department,
Dept. of Mechanical Engineering
ANITS, Sangivalasa,
Visakhapatnam.

PROJECT GUIDE

J.V.Bhanutej 28/5/22

Sri J.V.BHANUTEJ
Assistant Professor,
Dept. of Mechanical Engineering,
ANITS, Sangivalasa,
Visakhapatnam.

PROFESSOR & HEAD
Department of Mechanical Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES
Sangivalasa 531 162 VISAKHAPATNAM Dist A P

THIS PROJECT IS APPROVED BY THE BOARD OF EXAMINERS

INTERNAL EXAMINER

PROFESSOR & HEAD
Department of Mechanical Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES
Sangivalasa-531 162 VISAKHAPATNAM Dist A.P

EXTERNAL EXAMINER

ACKNOWLDGEMENT

We express immensely our deep sense of gratitude to **Mr. J. V BHANUTEJ**, Assistant Professor, Department of Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences, Sangivalasa, Bheemunipatnam Mandal, Visakhapatnam district for his valuable guidance and encouragement at every stage of the work made it a successful fulfillment.

We are very thankful to Professor **T. V. HANUMANTHA RAO**, Principal. **DR. B. NAGARAJU,** Head of the Department, Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences for their valuable suggestions.

We express our sincere thanks to the members of non-teaching staff of Mechanical Engineering for their kind co-operation and support to carry on work.

Last but not the least, we like to convey our thanks to all who have contributed either directly or indirectly for the completion of our work.

N.SUDHIN RAJ	(318126520195)
P.SRINIVAS	(318126520197)
Y.MURALI	(319126520L38)
P.GANESH	(319126520L37)
M.AKHIL	(319126520L50)

CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF GRAPHS

ABSTRACT

The Aerofoil section is the incarnation of a wing or a lifting surface which is very important in an airplane wing design. The design of airfoils is an important aspect of aerodynamics. Different airfoils are used for different flight regimes. While the shape of the Aerofoil changes, their aerodynamic characteristics also change. This project deals with a standard symmetrical Aerofoil as reference and the effect of changes in shape due to minor variations in the coordinates. New Aerofoil shapes have been produced in this optimization process. At zero angles of attack, asymmetric aero foils can create lift, but symmetric aero foils have no lift. The design of an airfoil has a significant impact on the lift produced to an aircraft as well as the thrust necessary to achieve that lift. The aerodynamic characteristic results such as the coefficients of lift and drag (Cd, Cl), pressure coefficient (Cp), are noted for three different profiles, produced from the standard NACA 23012, 23014, 63415 Aerofoil with changes in the chord thickness distance but no change in the maximum thickness in percentage of chord.

Design Modeler is used to create the geometry for the airfoil. NACA 23012, NACA 23014, and NACA 63415 are three kinds of airfoils that are examined. Ansys Fluent is used to do the study of the airfoil. Constant density is used to compute the coefficients of lift and drag across the segment. From 0 to 18 degrees AOA, the analysis is carried out. Using the NACA profiles a 3D model of flight wing is created in CATIA software. The modus-operandi used in this optimization process is the Computational Fluid Dynamics (CFD).

Flow changes have been recorded for these Aerofoil shapes and the results are arrived for finding the best Aerofoil that can be advisable to be used in compressors, turbines, etc. with reduced flutter and maximum life. A numerical CFD analysis of NACA profiles is undertaken in this work. ANSYS Fluent software is used to do the analysis. NACA23012, NACA 23014, and NACA 63415 are considered 2D profiles at first. The analysis is done up to 18 AOA to find the optimal aero foil. In tabular columns and graphs, the individual profile drag and lift coefficient are recorded.

CHAPTER – 1

INTRODUCTION

1.1 Motivation

Now days, rise in new kinds of technologies made humans to prefer luxurious life. From moving one place to another place we have to depend on transport. Generally, the modes of transport are road, air and water. To travel long distance with in short time, air transport is one of the source. So that domestic and international flights are the best option. The flight design depends upon the aerodynamic characters such as drag and lift. The Engine performance is increased when the drag force is reduced. The flights can travel from ground level to certain altitude. When the flight flying, the drag and lift forces will change. The analysis of these aerodynamic forces is depends on ground effect. This type of study is advanced and trending topic for researchers to design flight. For many years, researches were going on for the ground effect. Finite element analysis is a good option than the manual calculation and for experimental point of view. The flight is a complex shape to find out the aerodynamic characters of flight and numerical analysis is best option to reduce the time for solving and to obtain precise results. This type of study is helpful for flight manufacturing industries. Before constructing a flight, the drag acting on the flight is known before using CAD software and will give approximate results. Wing is one of the main parts of the flight. On these wings, aerodynamic forces with ground effect by using simulation software by considering different level of wing length to height ratio is planned in this work.

1.2 Background

There are two different types of flights are available these days, they are fixed wing flight and rotating wing flights. For long distance travelling and high speed, fixed wing flight has huge demand. But the flight controlling is mainly depending on the wing and wing pitch angle. For this type of flights, when the wing size and structure geometry increases, the drag force effect will be more. The resisting force is high that the engine performance is needed more than usual. Because of high usage of engine performance, the fuel cost increases and passenger ticket cost also increases. For this type of flights, the reduction of drag force is a big task for aircraft designers. There are various kinds of wings shapes that are available in the market with different series with NACA profile. When the flight wing's pitch angle changes it is necessary for the industries to check on the change in drag force. If the drag force is reduces, the engine performance is drastically improves and there would not be much use of engine power. So to identify the particular drag force, sensors are needed and real time set up is need and it is a highly complicated task for researcher and manufacturers. So here the alternative method called finite volume techniques is applied and analysis can be preceded. Because of this, working time is reduced, equipment's and other components cost can be avoided completely as the work is completely depending on the software. In the software, aerodynamic resistance force, dynamic force results are gained can be performed using software. Based on this, drag can be reduced for any geometric models can be formed.

1.3 Aerodynamic Forces on Aero foils

1.3.1 Flight

Any object under moving condition and stationary condition if the lifting forces are produced they are called lifting body. These lifting bodies are available and depend on size and shape. Airfoil is one of the lifting body. This airfoil body can be characterized by depending on lift and drag ratios. This airfoil should produce high amount of lift and lower amount drag, only then the body is known as efficient body. If airfoil moved in air, due to the air movement around the airfoil depends on the airfoil shape, a low pressure will create. At the top surface of the airfoil and high amount of pressure will create at lower surface of airfoil. Due to this pressure difference, the body can travel from high pressure to low pressure and body gains the lifting forces and airfoil is pushed toward upward direction to tilt back slightly.

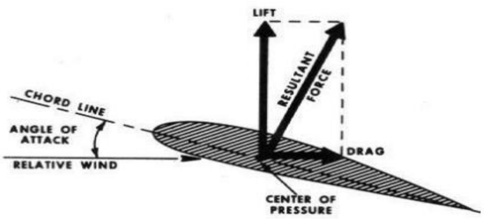


Fig1.3.1: Lift and Drag Forces Acting upon an Aero foil

Due to this motion two different forces will act on airfoil which are lift and drag force. The force which is perpendicular to wind direction is called lift force. The parallel direction is called drag force. Bernoulli's theory will be helpful to calculate how the flight will fly.

1.3.2 Lift

Lift is a force which is acting on airfoil and it can be calculated with non-dimensional parameter named lift coefficient.

$$C_L = \frac{L}{\frac{1}{2}\rho V^2 S}$$

Where Cl – coefficient of lift

L= length

ρ = density

V=volume

S= swept area

1.3.3 Drag

It is a main considerable parameter on airfoil and it can be characterized by two types which are skin friction drag and pressure drag.

Skin friction drag:

Skin friction drag is created by resulting from shear stresses acting on the airfoil body. It depends on the geometry and will remain constant with ground effect

1.3.4 Pressure Drag:

Pressure drag is equal to the rate of change of air particles' linear momentum normal to the local surface in a local surface's co-moving inertial frame minus pressure forces. The pressure drag is developed only when the pressure acting on airfoil profile while the air flows around the airfoil. This type of pressure drag is further classified into wave drag, induced drag and boundary layer pressure drag.

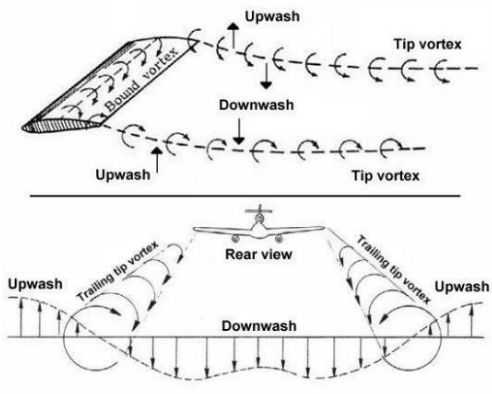


Fig1.3.4: The Development of Wingtip Vortices [2]

In pressure drag, induced drag is main factor to consider. The maximum forces available on airfoil in the above three types of drags. Higher pressure will act at the front and bottom of the airfoil. The lower pressure will act at tip of airfoil. Due to this, circular flow can show at tip of the wings. This circular flow of air at the tip of the wing and it further moves down. It is known as downwash. While the air is circulating, it will create small amount of lift force in to drag force. The drag force can be calculated with non-dimension parameter coefficient of drag.

$$C_D = \frac{D}{\frac{1}{2}\rho V^2 S}$$

Where CD – coefficient of drag

L= length

ρ = density

V=volume

S= swept area

1.3.5 Airfoil

It is a structure with curved surfaces designed to give the most favorable ratio of lift to drag in flight, used as the basic form of the wings, fins, and horizontal stabilizer of most aircraft. To the super critical airflow a separate airfoil is needed. In between 1960s to 1970s, National Aeronautics had developed a new model of airfoil. It is described as increase nose radius, flat upper surface, blunt tailing gate, modern supercritical airfoil and cambered rear. CFD package is more useful to develop these kinds of models. The purpose of designing of airfoil shape is to reduce the shock waves and drag force. After the model is developed, it gains attractive demand for commercial aircraft companies. Due to this, different types of models are developed by National Aeronautics

1.4 Problem Definition

Day by day, the advancement in technologies is improving. Based on these, people's mindset and comfort levels are also changing. To travel from one place to another in regular basis, the automobile vehicles, trains and flights plays a key role in transportation. To run such type of vehicles, it needs fuels or electricity. Using this input source, engine or motor are run and provides mechanical energy to provide forward and backward movement. When the flight moves, wind will resist that movement which is called drag force. Some lifting forces will induce due to the presence of air. When the flight wing structure is improper, the drag force will act high on the wing. Due to this, to overcome such effects, the engine performance should be more than usual. So the design of aircraft wings is an important factor for flight manufacturers. It depends on aerodynamic characteristics on Wind in Ground effect planes. In this work, aerodynamic analysis on NACA profile is planned through CFD. Lift force, drag force and pressure readings are planned to measure with the help of ground clearance.

1.5 Aim and Objective

The main aim of this work is to analyses the aerodynamic characteristics like drag, lift and pressure reading on aero foil. Once the wing's angle of attack is changes, how the lift force and drag force is changed and how to avoid the drag force on flight wing is a main aim to consider and further work is extended based on this. The study on importance of drag, lift coefficient and dynamic pressures are to be studied on real time model. According to previous articles, new geometry models have to be planned to reduce the resisting force on flight. The total project is depends upon ANSYS FLUENT software. So the main concentration is shown to simulation software named FLUENT. In this, different types of analysis are available but among them, steady state analysis is required to run such type of analysis. After performing analysis for all types of models, output result plotting is required to easily understand and helpful for comparison of results.

CHAPTER - 2

LITERATURE REVIEW

In this chapter, the details about previous research articles with regards to this work are discussed. In past few years, work related to airfoils for both numerical and experimental works were performed.

2.1 Numerical Work on Airfoil Wing

The numerical research has been the primary topic of ground effect analysis over the beginning of observing this occurrence. Saunders demonstrated contortions in numerical and experimental analysis under the ground effect. Over different models, he revealed that when the boundary condition is steady and unchanged, the numerical value and the experimental value will differ. He repeated exact experimental methods that maybe applied which are intended to validate the numerical values. The methods that are determined to be ahead of fixed ground plane based on him are lineal visualization with the towing technique and airfoil. In the end of 1970s some research studies were done base on fluid mechanics which is related to ground effect. In this study, the more focus is on the high Reynolds number with low Mach numbers. In this study, they worked in blunt body which is subjected to ground effect condition by considering various high to chord ratios. It is mainly concentrated on extreme and normal ground effect. In the extreme ground effect, lower lifting is acting on the wing surface due to the absence of air.

- **Coulliette.C et.al [1]** study states that steady state aero foil ground effect is studied both numerically and analytically. Discrete vortex and linear vortex panel methods are applied to a parabolic arc and symmetric Joukowski aero foil, respectively. A single vortex model for the flow over the parabolic arc aero foil is developed. The single vortex model and other analytical solutions, valid either near or far from the ground, are compared with the numerical results. The numerical results are used to delineate the influences of angle of attack, camber and thickness. For small values of camber and angle of attack, normalized lift is enhanced near the ground and reduced far from it. For a fixed distance above the ground, normalized lift decreases with increasing angle of attack and camber. Thickness reduces lift at all heights above the ground. The effect of camber, attacking angle, thickness of airfoil with ground effect is important study and many researches were done by Potkin. He concluded that lift forces increases as ground clearance decreases and also the lift forces increases with the increase in airfoil thickness. The vortices shed from airfoil in ground effect are a good study which was done in 2003 by Fonseca. The author discussed about the aerodynamic effect and pressure distribution with respect to ground.

- **Nuhait et.al [2]** performed the numerical analysis on steady and unsteady flow by considering the ground effect which was done in 1989. The aerodynamic coefficient was higher in unsteady flow when compared to steady flow. Based on this, some numerical approaches were performed by using unsteady flow by considering the ground effect. This research was performed while flight landing and takeoff positions and aerodynamic coefficients are studied.

- **Hsiun et.al [3]** had done work on NACA 4412 profile airfoil with the help of computational research. The CFD analysis is performed to analyse the effect of Reynolds number on airfoil which is operating in ground effect. For CFD analysis, PHEONICS code is used. In this work, turbulent model is considered with K-e turbulent model. Initially, the work is taken from experimental work which is done by Pinkerton. It was validated with computer numerical analysis. If the Reynolds number increases, lift force increases. When the distance between the ground and airfoil decreases, the drag force was decreased.

- **Steinbach et.al [4]**, had evaluated the work of Hsiun with steady ground plane and not considering the ground plane motion. In the study, he discussed about slip and no slip condition and also type of errors faced to solve Napier's equations.

- **Chang et.al [5]** used experimental work and CFD. The CFD analysis was performed to validate the experimental work. In this work, steady and moving ground plane effects on the airfoil is considered to obtain accurate aerodynamic forces as same as the experimental work. The lift force was same for two operating conditions and some difference was observed due to the drag force. The drag force was higher when the ground is in motion condition.
- **Barber et.al [6]** said that to study on the ground effect, ground plane should be in moving condition. In his study, four types of analysis was done with same wind

speed by considering the ground plane as stationary, moving, slip and no slip. Among all the four conditions, ground motion with same wind speed gives more accurate as same as the experimental work.

- **Saunders.G.H et.al [7]** said the vehicle aerodynamics concerns the effects arising due to motion of the vehicle through, or relative to, the air. Its importance to road vehicles became apparent when they started to achieve higher speeds. The automobile as we know it came onto the scene in the last decade of the nineteenth century. Its beginnings roughly coincided with the advent of powered flight, and perhaps for this reason, it became of interest to aerodynamicists right from the start. One of the first attempts to apply aerodynamic principles to road vehicles was the streamlining.

- **Tuck, E.O et.al [8]** study says that flow induced by a body moving near a plane wall is analyzed on the assumption that the normal distance from the wall of every point of the body is small compared to the body length. The flow is irrotational except for the vortex sheet representing the wake. The gap-flow problem in the case of unsteady motion is reduced to a nonlinear first-order ordinary differential equation in the time variable. In the special case of steady flow, some known results are recovered and generalized. As an illustration of the unsteady theory, the problem is solved of a flat plate falling toward the ground under its own weight, while moving forward at uniform speed.

2.2 Summary:

Hence by considering and studying the above journals we got an idea of how to use the different aero foil profiles. In this era we have got a clear idea on the manufacturing the required aero foil profile to maintain minimum drag and maximum lift. In this chapter, our detailed research on various research papers has been described. The inputs we took from these papers help us in designing and fabricating the model according to our requirements. These will be discussed in detail in subsequent chapters.

CHAPTER – 3
3D MODELLING OF AEROFOIL USING CATIA

Throughout the history of our industrial society, many inventions have been patented and whole new technologies have evolved. Perhaps the single development that has impacted manufacturing more quickly and significantly than any previous technology is the digital computer. Computers are being used increasingly for both design and detailing of engineering components in the drawing office.

Computer-aided design (CAD) is defined as the application of computers and graphics software to aid or enhance the product design from conceptualization to documentation. CAD is most commonly associated with the use of an interactive computer graphics system, referred to as a CAD system. Computer-aided design systems are powerful tools and in the mechanical design and geometric modeling of products and components. There are several good reasons for using a CAD system to support the engineering design

Function:

- To increase the productivity
- To improve the quality of the design
- To uniform design standards
- To create a manufacturing data base
- To eliminate inaccuracies caused by hand-copying of drawings and

inconsistency between

- Drawings

3.1 CAD Software

Software allows the human user to turn a hardware configuration into a powerful design and manufacturing system. CAD/CAM software falls into two broad categories, 2-D and 3-D, based on the number of dimensions are called 2-D representations of 3-D objects is inherently confusing. Equally problem has been the inability of manufacturing personnel to properly read and interpret complicated 2-D representations of objects. 3-D software permits the parts to be viewed with the 3-D planes-height, width, and depth-visible. The trend in CAD/CAM is toward

3-D representation of graphic images. Such representation approximates the actual shape and appearance of the object to be produced; therefore, they are easier to read and understand.

3.1.1 CATIA

Initially, CATIA name is an abbreviation for Computer Aided Three-dimensional Interactive Application the French Dassault Systems is the parent company and IBM participates in the software's and marketing, and CATIA is invades broad industrial sectors, and has been explained in the previous post position of CATIA between 3d modeling software programs.

A window will be opened and there are types of design.

Types are:

1) Layout
2) Sketch
3) Part
4) Assembly
5) Manufacturing
6) Drawing
7) Format
8) Report
9) Diagram

When clicking on new we get a window and select part – solid- ok. There are few of sub type in part.

1) Solid
2) Sheet metal
3) Bulk

Here a display will be shown in the CATIA and the three planes will be shown here.

1) Front
2) Top
3) Right

In the left side of the CATIA we find the Model tree. This model tree is shown in the planes. The design could be saved as PRT0001.PRT in the model tree.

On top of the CATIA display we find the menu bar. In that menu bar we find

- Model
- Analysis
- Annotate
- Render
- Tools
- View
- Flexible modeling
- Applications

In model we have different tools to select. The tools which we have in model is

❖ Operations
(a) Regenerate
(b) Copy
(c) Paste
(d) Delete

❖ Get data
(a) User defined feature
(b) Copy geometry
(c) Shrink wrap

❖ Datum
(a) Plane
(b) Axis
(c) Coordinate system
(d) Sketch

❖ Sketch based features
(a) Extrude
(b) rib
(c) Revolve
(d) Multi section solids

❖ Dress up features
(a) Hole
(b) Round
(c) Chamfer
(d) Draft
(e) Shell

❖ Transformation features
(a) Pattern
(b) Mirror
(c) Trim
(d) Merge
(e) Extend
(f) Offset
(g) Intersect
(h) Project

(i) Thicken
(j) Solidify

❖ Surfaces
(a) Boundary blend
(b) Fill
(c) Style
(d) Free style

❖ Model intent
(a) Component interface

From the above tools we are firstly going to use the datum tool. In datum tool we are going to choose sketch. Before using sketch select the plane of the CATIA display and then go to sketch. So that generating of face can be done in CATIA. Draw the drawing which is having an accurate dimension then convert to three dimensional solid. Airfoil NACA 23012 and NACA 23014 series are used to modeled the YAK 54in CATIA, was used to design the aircraft wing according to specifications given below. The fully designed model was saved as a file with extension ‘.igs.

Table 3.1.1 Yak-54 Aircraft Wing Lifting Surface Dimension

S	1012643 mm^2
b	2668 mm
ĉ	1.45
λ	.46
AR	5.77
cr	560mm
ct	266mm

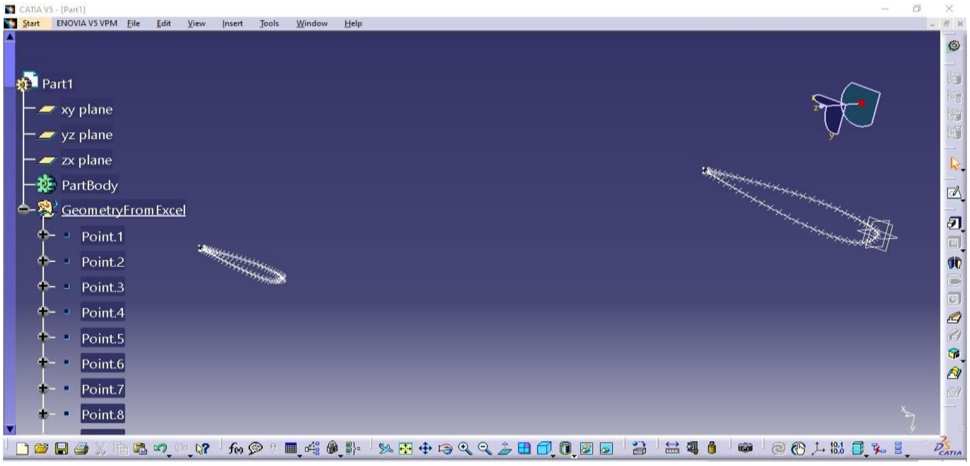


Fig 3.1.1: sketch view of Wing model.

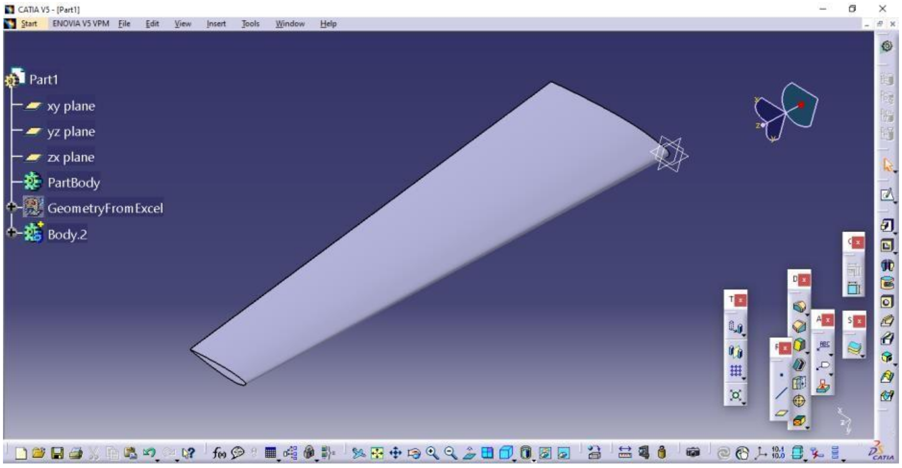


Fig 3.1.2. Wing Section of YAK 54 NACA 23012 at tip and 23014 at root of aero foil

Wing is a main component in Flight. Here Wing is directly attached to Fuselage. CATIA software is used to create it. All dimensions are taken in mm. sketch is created using Macro concept in Excel.

CHAPTER - 4
COMPUTATIONAL FLUID DYNAMICS

A computation fluid dynamic analysis on NACA 4412 airfoil model is studied in this work; the geometrical parameters of airfoil wing are taken from National Aeronautics Once the geometry model is done, the aerodynamic simulation is done to find out the drag and lift coefficient. Here, 2D wing are taken and simulation is done by changing the angle of attack of the flow. The simulation software is trending now days in various industries which are mainly construction, shipping, automobile etc. This simulation software is developed by different companies depending on CAD. The companies are choosing and purchasing depending on cost, services and maintenance. But this software can be classified into three categories such as drafting, 3D modeling design and analysis software. In this work, ANSYS software is used.

4.1 Benefits of CFD

To solve experimentally for fluid flow related problem is a complicated task. To solve such problems, various kinds of sensors are required and at the same time air cannot be seen. But in CFD it is possible. To test experimentally on airfoil, forming geometry is complex work and also to find out the aerodynamic characters such as to detect lift and drag is required many sensing sensors. This will cause huge burden to the companies. So CFD is one of the sources to provide best results.

4.2 CFD Process

From the beginning of geometry till gaining results; there are different types of step included such as

1. Geometry formation related to the current work.
2. Applying meshing to the selected geometry.
3. Defining the materials
4. Choosing type of study either steady state or transient
5. Assigning material properties to cell zones
6. Applying boundary condition at inlet and outlet.
7. Requesting the output reports

8. Initializing the process
9. Run the analysis
10. Gaining output results such as pressure, velocity, lift, drag and moments.

Initially, a 2 dimensional simulation is planned on airfoil so that NACA 23012 profile geometry is selected. To prepare the geometry, it needs x,y and z coordinate data. These data are gathered from National Aeronautics. The data is initially in the form of x, y. The preparing of geometry is planned in ANSYS software. To import the data in terms of points, it should be set of format such as group, point number, x, y and z direction coordinates. Once entered into geometry, there is 3D point option in the geometry. Through this option data can be imported with set of point. Using this data points, a 3D curve is prepared. The profile chord length is 1m and maximum thickness of airfoil is 12%.

4.2.1 Geometric Model and Air Domain

Once the airfoil is formed in closed boundary, it is converted to surface. Air domain geometry is critical to gain structure type of mesh. The front portion of airfoil is created as semicircular arc and after that lines are prepared at rectangular shape. The air domain length is taken as 20 times of chord length. The height of the airfoil is 12.5 times of airfoil chord length. The profile is closed and made in to surface. Using Boolean operation, subtract option and airfoil surface is subtracted from air domain boundary which is shown in the below image.

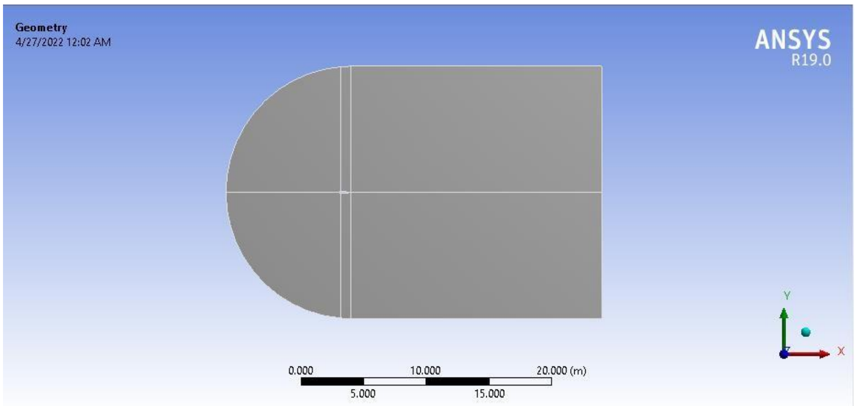


Fig 4.2.1: geometry model and air domain.

4.2.2 Structured Mesh on Airfoil Domain

Once the geometry is done, the next step is meshing. Meshing is an important consideration in this work. Different shapes of element types are available in the software. Depending on processing time, accurate result, the element shape must be selected. The main concentration is on the around airfoil boundary shape. Using these considerations, the meshing is planned. In this structured mesh, meshing is done to form higher order quadratic element. Initially edge meshing is done to prepare mesh planning and bios control is used with the factor of 20 because to place maximum concentration on airfoil bounded area.

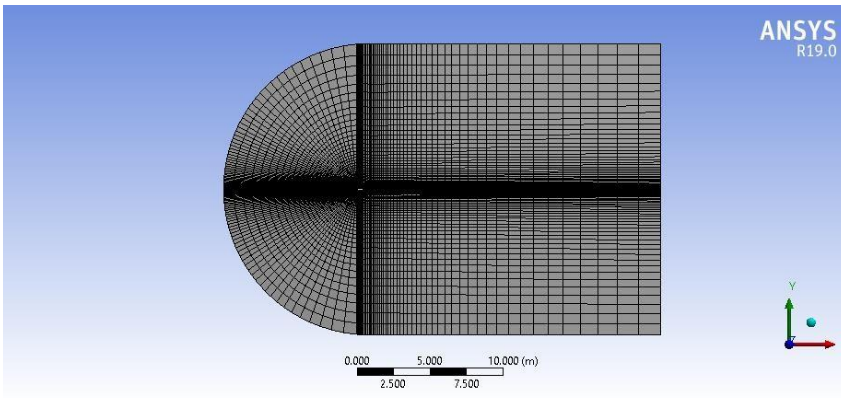


Fig 4.2.2: structured mesh on airfoil domain.

4.2.3 Aero foil With Detail View

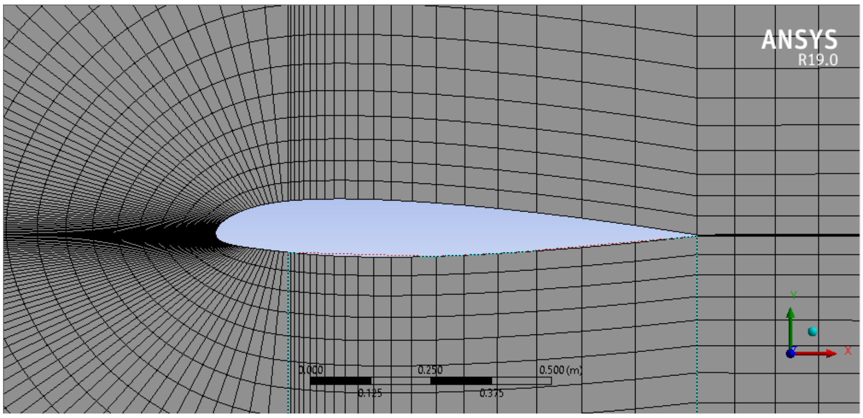


Fig 4.2.3: Aero foil with detail view.

Once the planning of meshing is done, assigning the boundary is an important task. In these boundaries, inlet name is given to send the air flow and the outlet name is given to send the air exist from the boundary. The airfoil wall name is given to airfoil region because the main work is to gain the aerodynamic results around the airfoil. The lower part of the boundary is named as ground and the upper part of the boundary is named as free surface.

4.2.4 Boundary Conditions on Air Domain

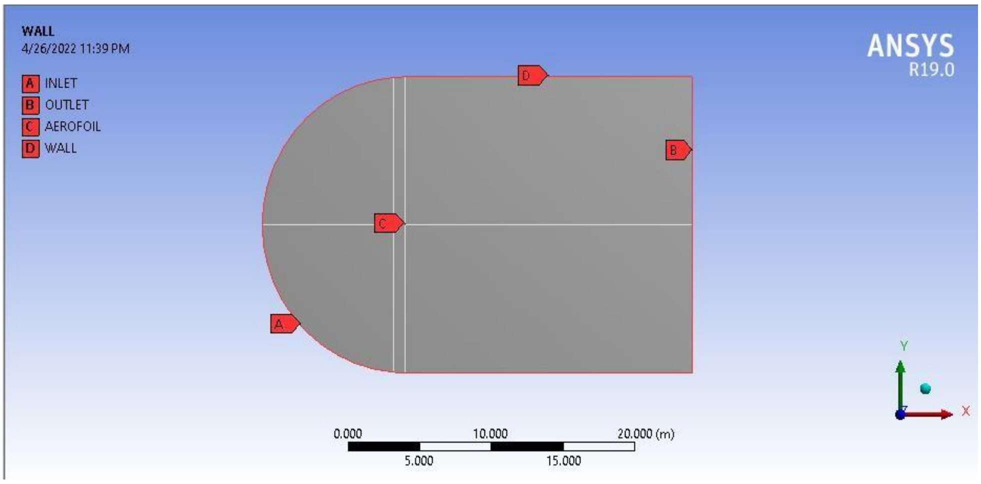


Fig 4.2.4: Boundary conditions on air domain.

Once meshing and naming for boundary condition is done, fluid simulation is the next procedure. They are various kind of software available to performed fluid simulation. Simulation software such as ANSYS FLUENT, ANSYS CFX, OPENFORM are available in the market. Depending on the availability ANSYS FLUENT software is selected. In this software, 2D simulation is performed. This software is divided into general, model, cell zones, material, boundary condition, report, report request, initialization and run analysis.

4.2.5 Turbulent Model

Once entered in FLUENT software, in general; pressure based 2D planner is selected. There is no gravity consideration is taken. The pressure based solver is considered due to the drag and lift produced due pressure variation around the airfoil. For other tasks, gravity is important. But for this particular problem there is no requirement of gravity. The 2D problems are characterized into two types. They are axis-symmetrical model and planner mode. In axis-symmetric, round shaped air domain section is taken for the analysis. But here, full model is considered from front portion of the airfoil to outside of the airfoil as air domain and not considering the axis. The domain is taken as planner.

In the model portion, turbulent model option is selected and energy is not taken as consideration as this work is not related to thermal problem. Different types of turbulent model available in the software. Depending upon the accurate results are obtained at vertices points and processing time. In this analysis, K-E turbulence model provides good agreement with experimental results.

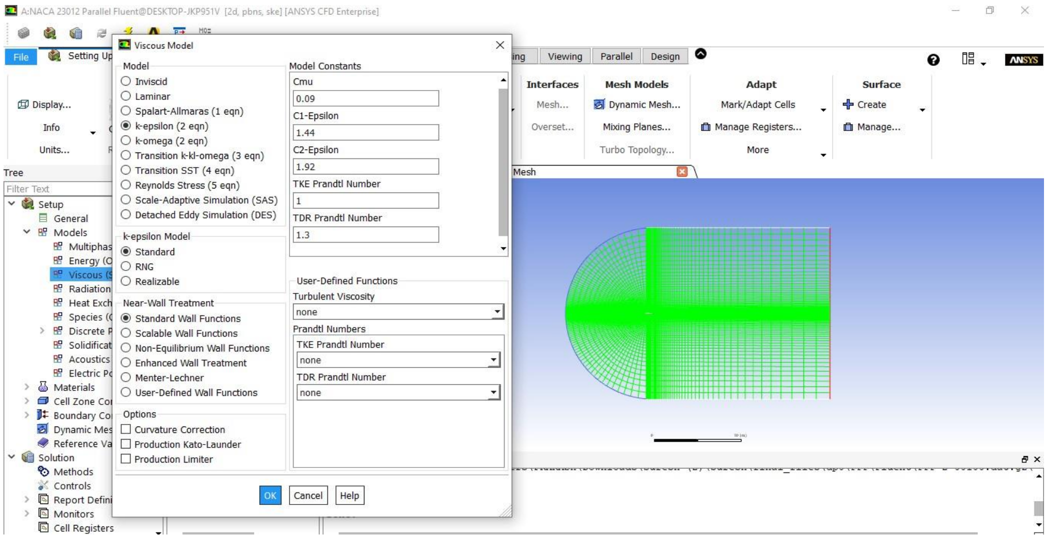


Fig 4.2.5: Turbulent model selected for analysis.

The air is taken as a material of the fluid and velocity of 30m/s is given to the inlet condition which is taken as constant throughout the entire work. Using this velocity coefficient of lift and drag is also calculated. The outlet boundary condition is taken as pressure outlet. Here the pressure value is 0 Pa applied at the pressure outlet condition. The software will work on the pressure difference; in inlet velocity is given that is why there will be more pressure at the inlet region. The negligible pressure is place at the outlet. Using these differences, air flows from inlet to outlet. The airfoil wall is places between inlet and outlet, the air will hit the foil, through this, and pressure variations along the boundaries are induced. Using report option in FLUENT we can gain the pressure results. In reference condition, reference area, density, velocity, length and pressure values are entered. These parameters will affect aerodynamic forces. At the inlet, initialization is started and the problem is solved in steady state condition pressure. Nearly 500 iterations were run. As there is structured meshed, at 100 iterations, the problem is solved. Once these steps are completed, problem must converge. To converge, for pressure, moment and velocity, convergence criteria of 10^{-5} is given to obtain accurate results. The convergence criteria mean error residuals between two iterations.

4.3 Results of Flight Model (Distributions)

The simulation was done by considered the ground motion. The bottom wall of the air domain is named as ground. The problem is steady state problem, the wind velocity and ground moving wall velocity is taken same as 30m/s. The pressure outlet of the domain is taken as 0. As same as 2D analysis, drag force and lift force for various ground clearance distance are analyzed. The problem is solved until iterated reached the convergence 10^{-5}.

4.3.1 Distributions of NACA 23012 at 0 AOA

4.3.1.1 Velocity Distribution

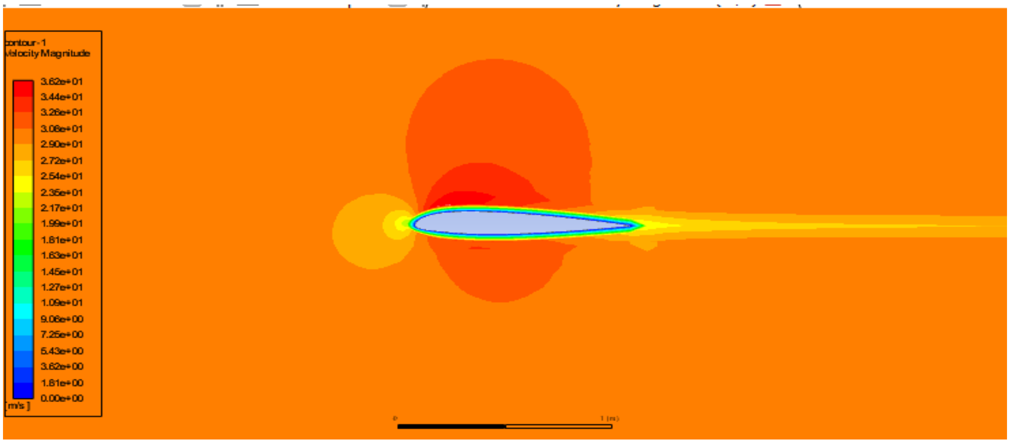


Fig 4.3.1.1: velocity distribution around airfoil NACA 23012 at 0 AOA.

The image above is velocity distribution around airfoil at 0 AOA is shown above. The red color in the image indicates maximum velocity distribution and the blue color indicates minimum velocity distribution. The maximum velocity distribution is observed at the top surface of the airfoil. The minimum velocity distribution is observed at the tip end of the air foil. When the air flows through the air foil, the velocity is high at the top and after the tip end of the airfoil, the turbulence flows through it continuously. The maximum velocity value is 3.82e+01 and the minimum velocity value is 0.00e+00.

4.3.1.2 Pressure Distribution

The pressure distribution around airfoil at 0^{o} angle of attack is shown in the image above. The maximum pressure distribution is displayed in red color, whereas the minimum pressure distribution is displayed in blue color. From the image, the maximum velocity is noticed at the starting tip of the airfoil where the pressure distribution value is 3e+02.

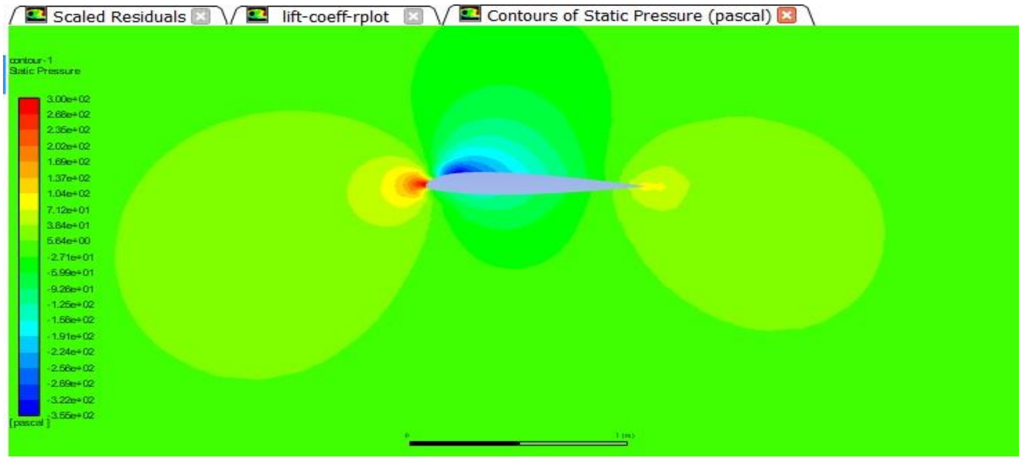


Fig 4.3.1.2: pressure distribution around aero foil NACA 23012 at 0 AOA.

The minimum pressure distribution is noticed at the top surface of the airfoil at 0^{o} angle of attack and the minimum pressure value is -3.85 e+02. When the air passes through the airfoil at 0 AOA, the pressure moved up throughout the top surface of the airfoil. Which means, the air foil moves upward at 0^{o} angle of attack

4.3.2 Distributions of NACA 23012 at 18 AOA

4.3.2.1 Velocity Distribution

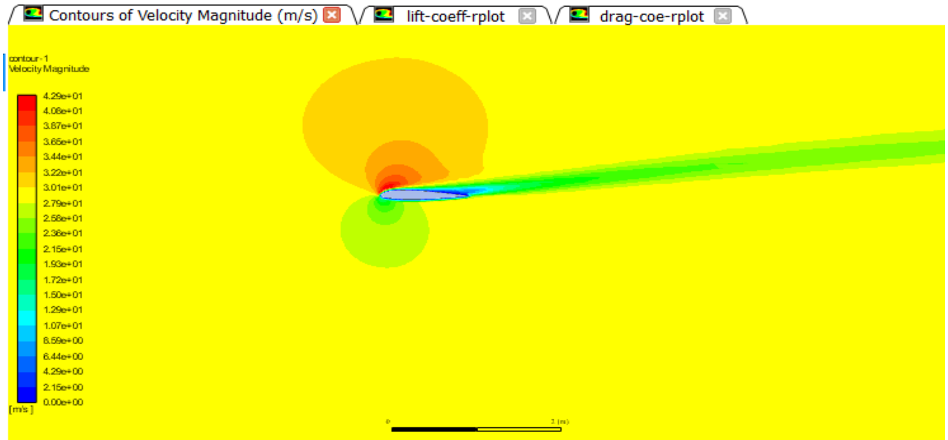


Fig 4.3.2.1: velocity distribution around airfoil NACA 23012 at 18 AOA.

The velocity distribution on airfoil at 18 AOA is observed at above image. The maximum velocity displayed in red color which is at half of the top surface. The minimum velocity displayed in blue color and it is noticed at the bottom tip of the airfoil and tip end of the airfoil. When the air flows through the airfoil, the velocity is moves up and the slowly turbulence reaches the end of the airfoil tip and moves continuously. The maximum velocity value is 4.26 e+01. The minimum velocity value is 0.00e+00.

4.3.2.2 Pressure Distribution

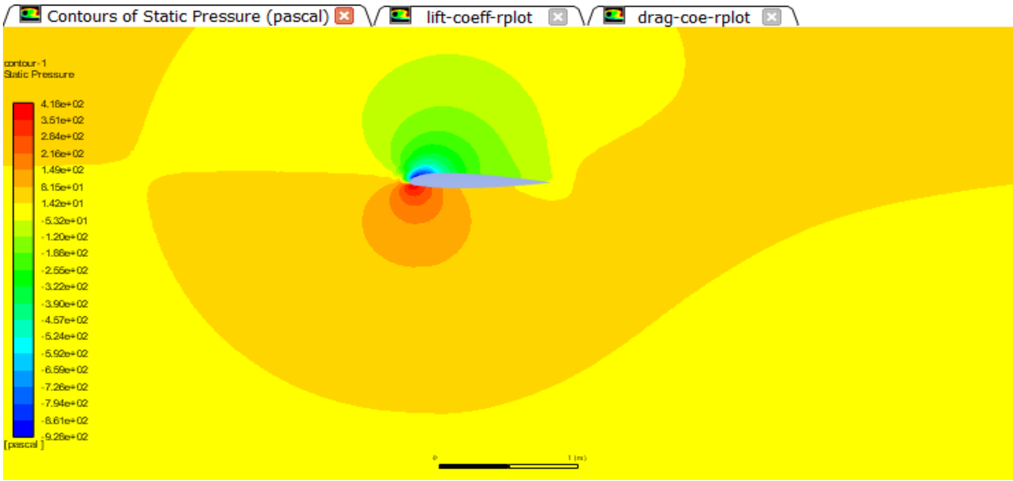


Fig 4.3.2.2: pressure distribution around aero foil NACA 23012 for 18 AOA.

The image above represents the pressure distribution around airfoil at 18^{o} angle of attack. The red color in the images indicates the maximum pressure distribution; whereas the blue color in the image indicates minimum pressure distribution. The maximum velocity is noticed at the bottom front tip of the airfoil where the pressure distribution value is 4.16 e+02. The minimum pressure distribution is noticed at the top front tip surface of the airfoil at 18^{o} angle of attack and the minimum pressure value is -9.29e+03. When the air flows through the airfoil at 18 AOA, the pressure slightly moved up at the starting top surface of the airfoil. That means, the air foil moves slightly upward and holds at 18^{o} angle of attack.

4.3.3 Distributions of NACA 23014 at 0 AOA

4.3.3.1 Velocity Distribution

The image above is velocity distribution around airfoil at 0 AOA is shown above. The red color in the image indicates maximum velocity distribution and the blue color indicates minimum velocity distribution.

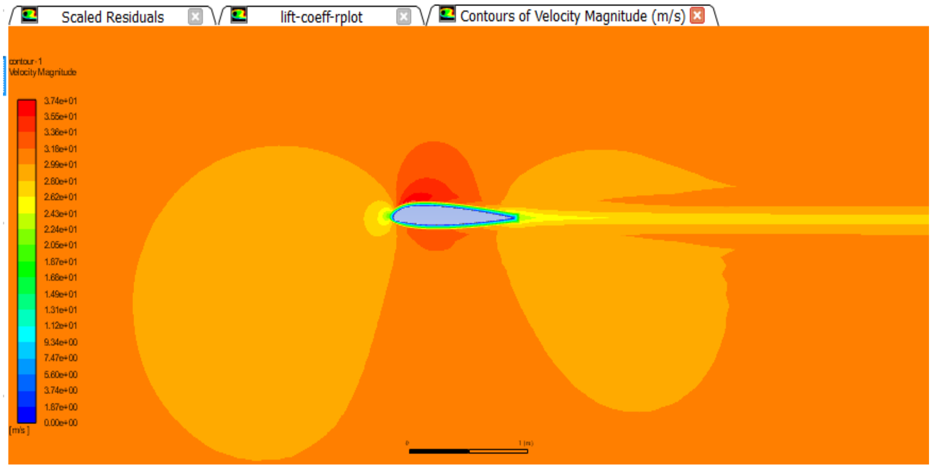


Fig 4.3.3.1: velocity distribution around airfoil NACA 23014 at 0 AOA.

The maximum velocity distribution is observed at the top surface of the airfoil. The minimum velocity distribution is observed at the tip end of the air foil. When the air flows through the air foil, the velocity is high at the top and after the tip end of the airfoil, the turbulence flows through it continuously. The maximum velocity value is 3.37e+01 and the minimum velocity value is 0.00e+00.

4.3.3.2 Pressure Distribution

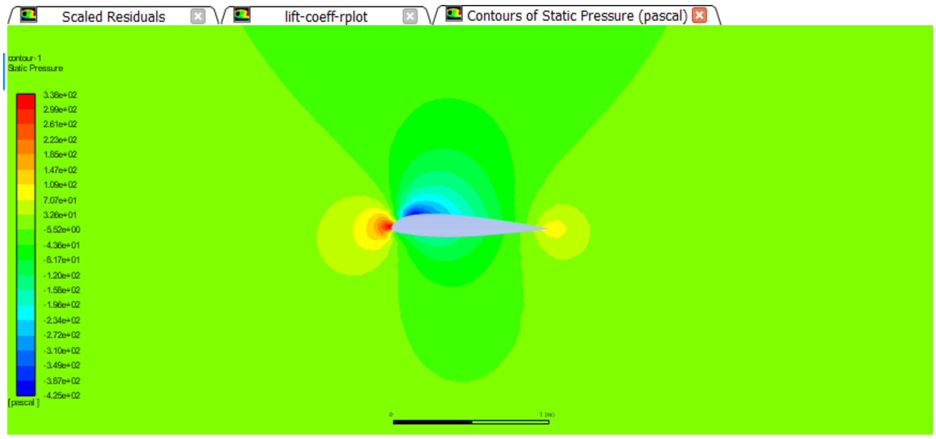


Fig 4.3.3.2: pressure distribution around aero foil NACA 23014 at 0 AOA.

The pressure distribution around airfoil at 0^{o} angle of attack is shown in the image above. The maximum pressure distribution is displayed in red color, whereas the minimum pressure distribution is displayed in blue color. From the image, the maximum velocity is noticed at the starting tip of the airfoil where the pressure distribution value is 3.96e+02. The minimum pressure distribution is noticed at the top surface of the airfoil at 0^{o} angle of attack and the minimum pressure value is -

4.25 e+02. When the air passes through the airfoil at 0 AOA, the pressure moved up throughout the top surface of the airfoil. Which means, the air foil moves upward at 0^{o} angle of attack.

4.3.4 Distributions of NACA 23014 at 18 AOA

4.3.4.1 Velocity Distribution

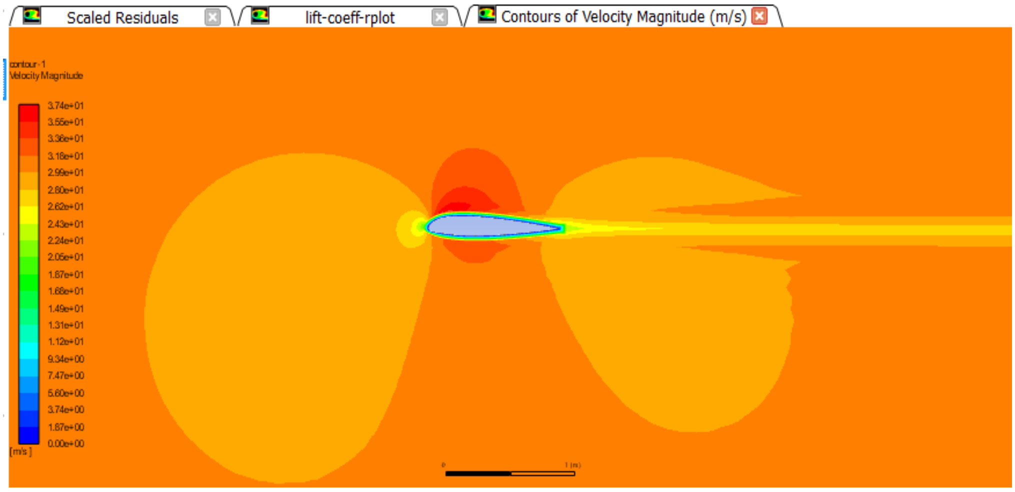


Fig 4.3.4.1: velocity distribution around airfoil NACA 23014 at 18 AOA.

The velocity distribution on airfoil at 18 AOA is observed at above image. The maximum velocity displayed in red color which is at half of the top surface. The minimum velocity displayed in blue color and it is noticed at the bottom tip of the airfoil and tip end of the airfoil. When the air flows through the airfoil, the velocity is moves up and the slowly turbulence reaches the end of the airfoil tip and moves continuously. The maximum velocity value is 3.7 e+01. The minimum velocity value is 0.00e+00

4.3.4.2 Pressure Distribution

The figure 4.3.4.2 represents the pressure distribution around airfoil at 18^{o} angle of attack. The red color in the images indicates the maximum pressure distribution; whereas the blue color in the image indicates minimum pressure distribution. The maximum velocity is noticed at the bottom front tip of the airfoil where the pressure distribution value is 4.4 e+02. The minimum pressure distribution is noticed at the top front tip surface of the airfoil at 18^{o} angle of attack and the minimum pressure value is 0. When the air flows through the airfoil at 18 AOA, the pressure slightly moved up at the starting top surface of the airfoil. That means, the air foil moves slightly upward and holds at 18^{o} angle of attack.

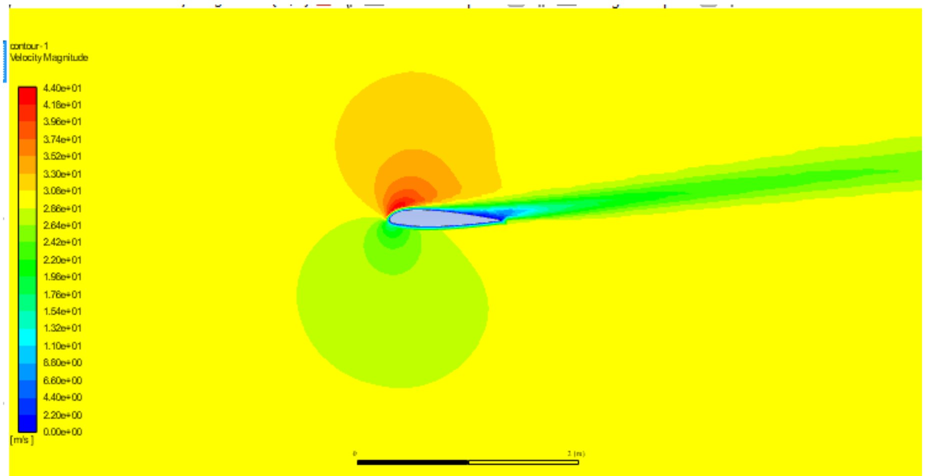


Fig 4.3.4.2: pressure distribution around aero foil NACA 23014 for 18 AOA.

4.3.5 Distributions of NACA 63415 at 0 AOA

4.3.5.1 Velocity Distribution

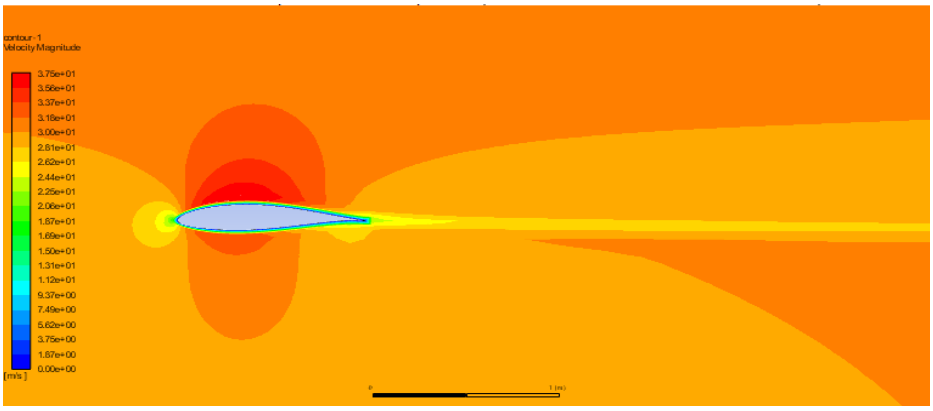


Fig 4.3.5.1: velocity distribution around airfoil NACA 63415 at 0 AOA.

The velocity distribution on airfoil at 0 AOA is observed at above image. The maximum velocity displayed in red color which is at half of the top surface. The minimum velocity displayed in blue color and it is noticed at the bottom tip of the airfoil and tip end of the airfoil. When the air flows through the airfoil, the velocity is moves up and the slowly turbulence reaches the end of the airfoil tip and moves continuously. The maximum velocity value is 3.76 e+01. The minimum velocity value is 0.00e+00.

4.3.5.2 Pressure Distribution

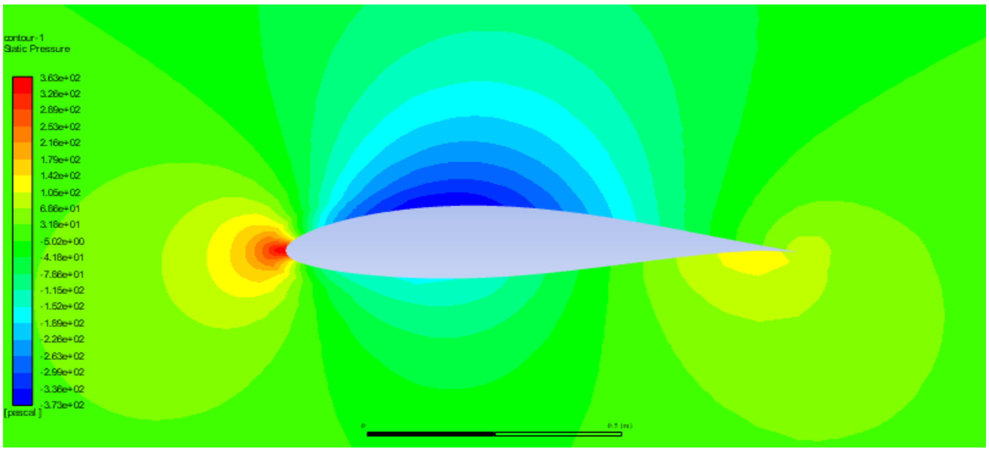


Fig 4.3.5.2: pressure distribution around aero foil NACA 63415 at 0 AOA.

The pressure distribution around airfoil at 0^o angle of attack is shown in the image above. The maximum pressure distribution is displayed in red color, whereas the minimum pressure distribution is displayed in blue color. From the image, the maximum velocity is noticed at the starting tip of the airfoil where the pressure distribution value is 3.63e+02. The minimum pressure distribution is noticed at the top surface of the airfoil at 0^o angle of attack and the minimum pressure value is – 3.72e+02. When the air passes through the airfoil at 0 AOA, the pressure moved up throughout the top surface of the airfoil. Which means, the air foil moves upward at 0^o angle of attack.

4.3.6 Distributions of NACA 63415 at 18 AOA

4.3.6.1 Velocity Distribution

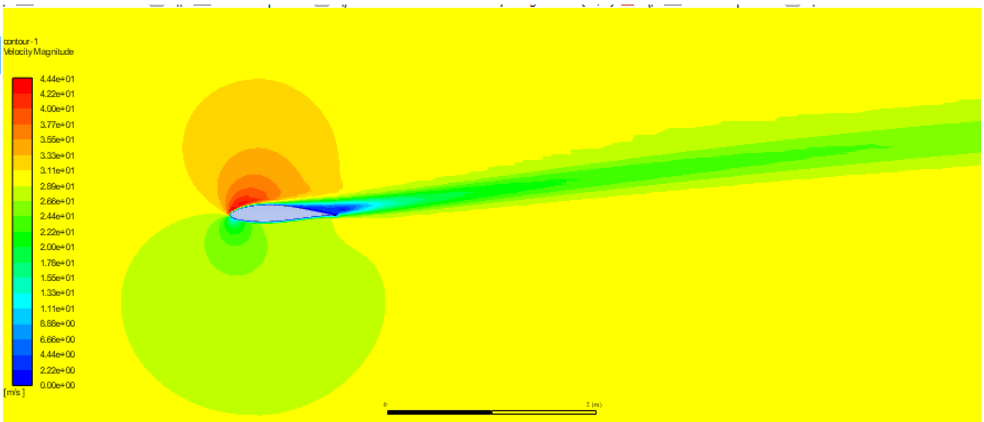


Fig 4.3.6.1: velocity distribution around airfoil NACA 63415 at 18 AOA.

The velocity distribution on airfoil at 18 AOA is observed at above image. The maximum velocity displayed in red color which is at half of the top surface. The minimum velocity displayed in blue color and it is noticed at the bottom tip of the airfoil and tip end of the airfoil. When the air flows through the airfoil, the velocity is moves up and the slowly turbulence reaches the end of the airfoil tip and moves continuously. The maximum velocity value is 4.44 e+01. The minimum velocity value is 0.00e+00.

4.3.6.2 Pressure Distribution

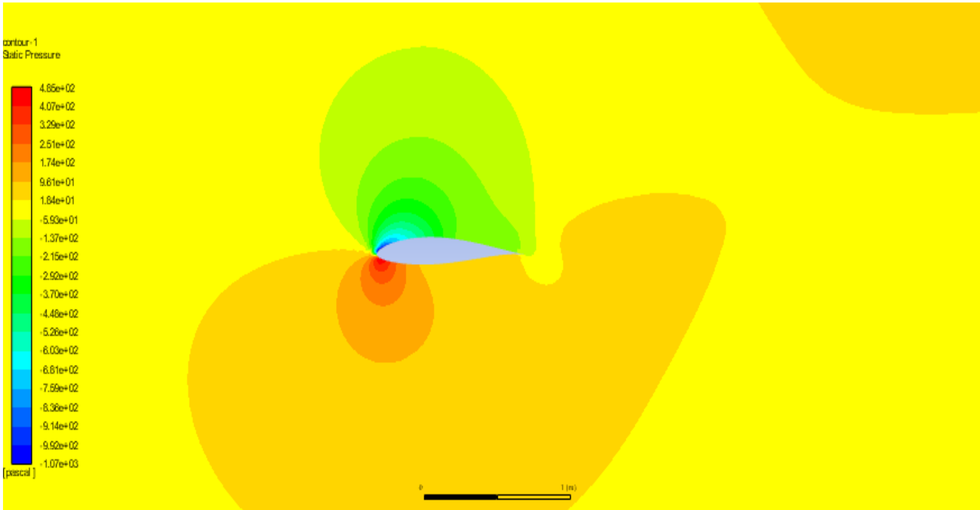


Fig 4.3.6.2: pressure distribution around aero foil NACA 63415 at 18 AOA.

The pressure distribution around airfoil at 18^o angle of attack is shown in the image above. The maximum pressure distribution is displayed in red color, whereas the minimum pressure distribution is displayed in blue color. From the image, the maximum velocity is noticed at the starting tip of the airfoil where the pressure distribution value is 4.86e+02. The minimum pressure distribution is noticed at the top surface of the airfoil at 18^o angle of attack and the minimum pressure value is -1.07 e+03. When the air passes through the airfoil at 18 AOA, the pressure moved up throughout the top surface of the airfoil. Which means, the air foil moves upward at 0^o angle of attack.

4.4 Results of Flight Model (Coefficients)

4.4.1 Drag Coefficient Plot for NACA 23012

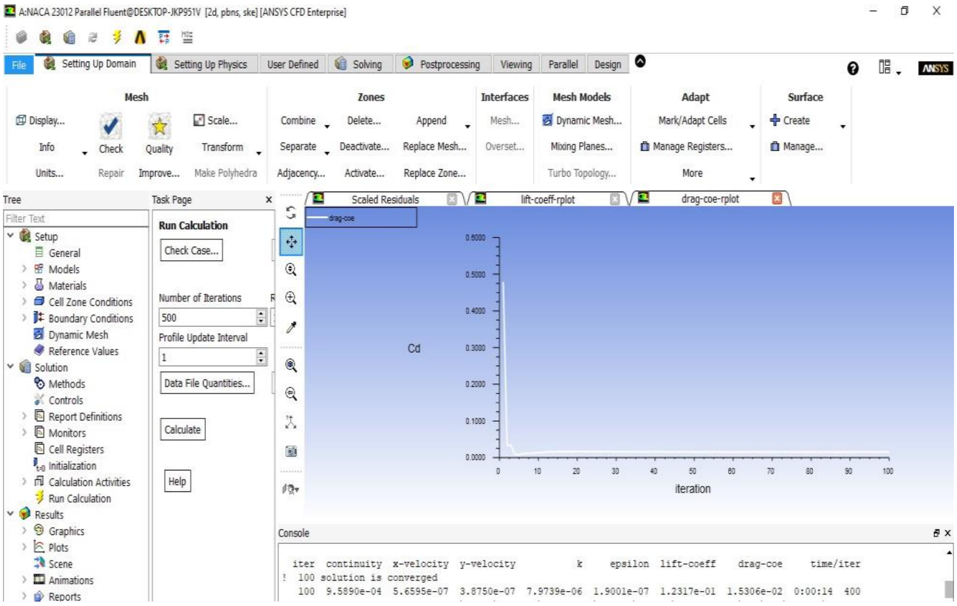


Fig 4.4.1: Drag coefficient plot for NACA 23012

Table 4.4.1: Drag coefficient plot for NACA 23012

AOA	NACA 23012
0	0.02
2	0.01
4	0.01
6	0.00
8	-0.01
10	-0.03
12	-0.04
14	-0.05
16	-0.05
18	-0.06

The above image shows the drag coefficient plot for NACA 23012. In this plot the x axis shows iteration number and y axis shows drag coefficients. At the starting of the processing time, air is hitting the flight and will create turbulence and a curve fluctuation is observed at the beginning of few iterations. After some iteration, it becomes idle. It continues till the problem is converged. The coefficient of drag reading 0.015 is noted.

4.4.2 Lift Coefficient Plot for NACA 23012

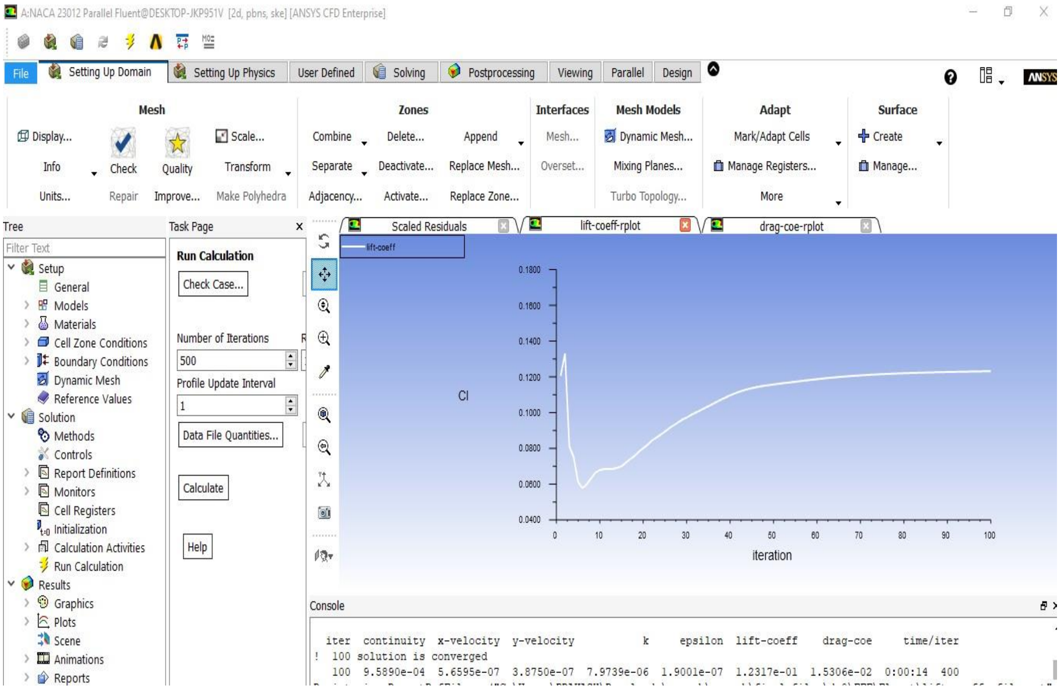


Fig 4.4.2: Lift coefficient plot for NACA 23012

Table 4.4.2: Lift coefficient plot for NACA 23012

AOA	ACA 23012
0	0.12
2	0.23
4	0.33
6	0.42
8	0.51
10	0.59
12	0.65
14	0.70
16	0.74
18	0.75

The lift coefficient plot for NACA 23012 is shown in the above image. In this plot the x axis represents iteration number and y axis represents lift coefficients. At the starting of the processing time, air is hitting the flight and will create turbulence and a curve fluctuation is observed at the beginning of few iterations. After some iteration, it becomes idle and proceeds till the problem is converged. The coefficient of lift reading 0.12 is noted.

4.4.3 Drag Coefficient Plot for NACA 23014

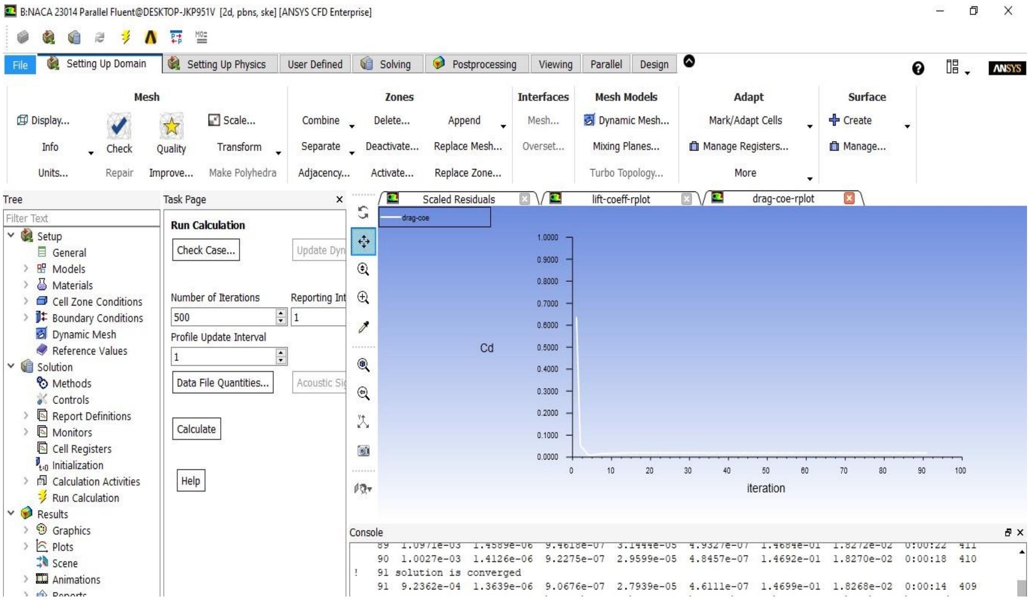


Fig 4.4.3: Drag coefficient plot for NACA 23014

Table 4.4.3: Drag coefficient plot for NACA 23014

AOA	NACA 23014
0	0.02
2	0.01
4	0.01
6	0.00
8	-0.01
10	-0.03
12	-0.04
14	-0.05
16	-0.06
18	-0.06

The above image shows the drag coefficient plot for NACA 23014. In this plot the x axis shows iteration number and y axis shows drag coefficients. At the starting of the processing time, air is hitting the flight and will create turbulence and a curve fluctuation is observed at the beginning of few iterations. After some iteration, it becomes idle. The coefficient of drag reading 0.015 is noted.

4.4.4 Lift Coefficient Plot for NACA 23014

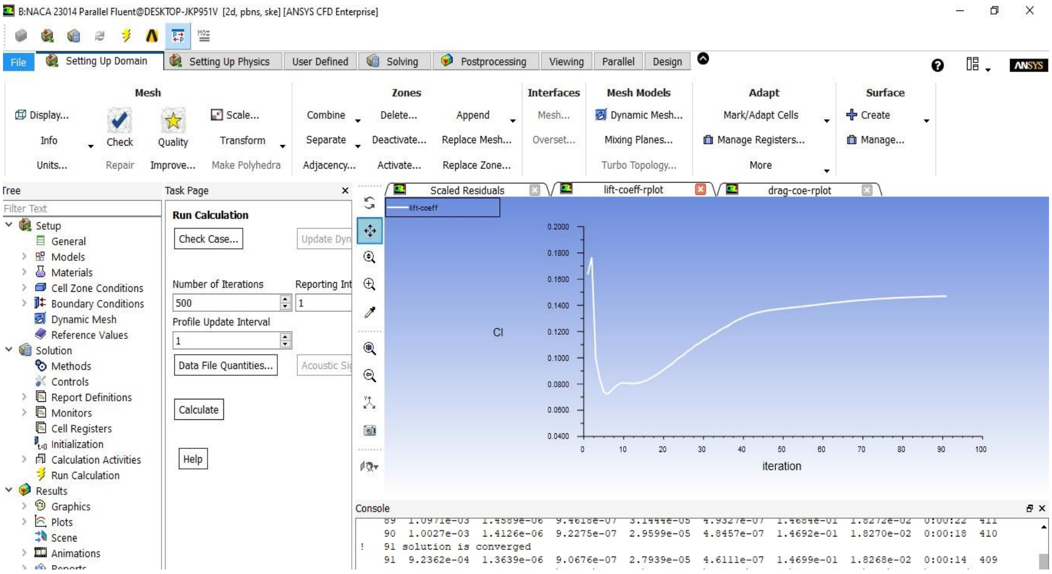


Fig 4.4.4: Lift coefficient plot for NACA 23014

Table 4.4.4: Lift coefficient plot for NACA 23014

AOA	NACA 23014
0	0.15
2	0.25
4	0.34
6	0.44
8	0.52
10	0.60
12	0.66
14	0.71
16	0.74
18	0.76

The lift coefficient plot for NACA 23014 is shown in the above image. In this plot the x axis represents iteration number and y axis represents lift coefficients. At the starting of the processing time, air is hitting the flight and will create turbulence and a curve fluctuation is observed at the beginning of few iterations. After some iteration, it becomes idle and proceeds till the problem is converged. The coefficient of lift reading 0.12 is noted.

4.4.5 Drag Coefficient Plot for NACA 63415

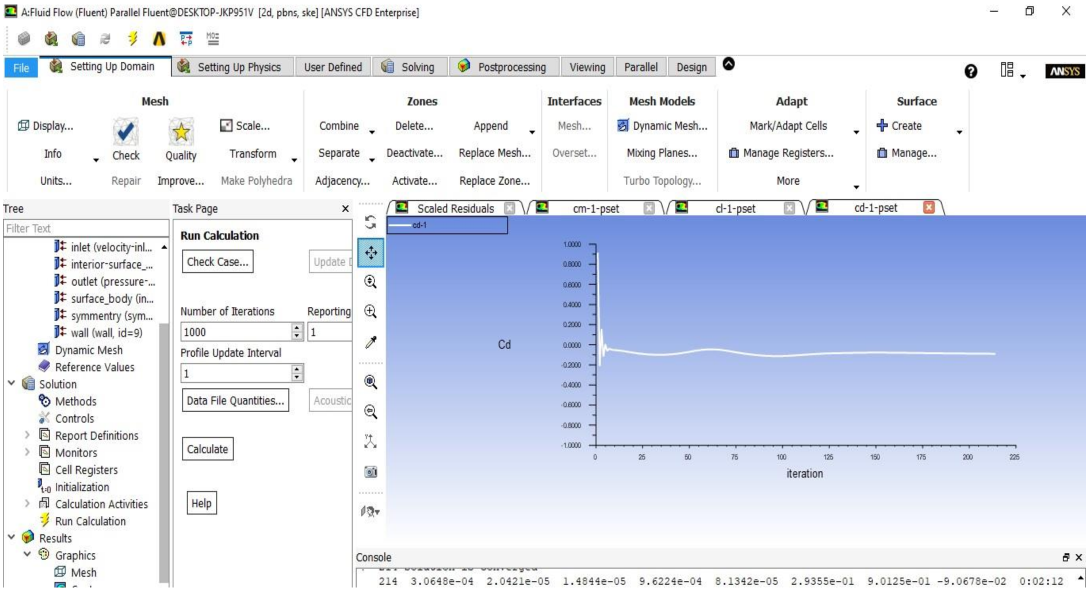


Fig 4.4.5: Drag coefficient plot for NACA 63415

Table 4.4.6: Lift coefficient plot for NACA 63415

AOA	NACA 63415
0	0.02
2	0.01
4	0.00
6	-0.02
8	-0.03
10	-0.05
12	-0.07
14	-0.08
16	-0.09

The above image shows the drag coefficient plot for NACA 63415. In this plot the x axis shows iteration number and y axis shows drag coefficients. At the starting of the processing time, air is hitting the flight and will create turbulence and a curve fluctuation is observed at the beginning of few iterations. After some iteration, it becomes idle. It continues till the problem is converged. The coefficient of drag reading -0.09 is noted.

4.4.6 Lift Coefficient Plot for NACA 63415

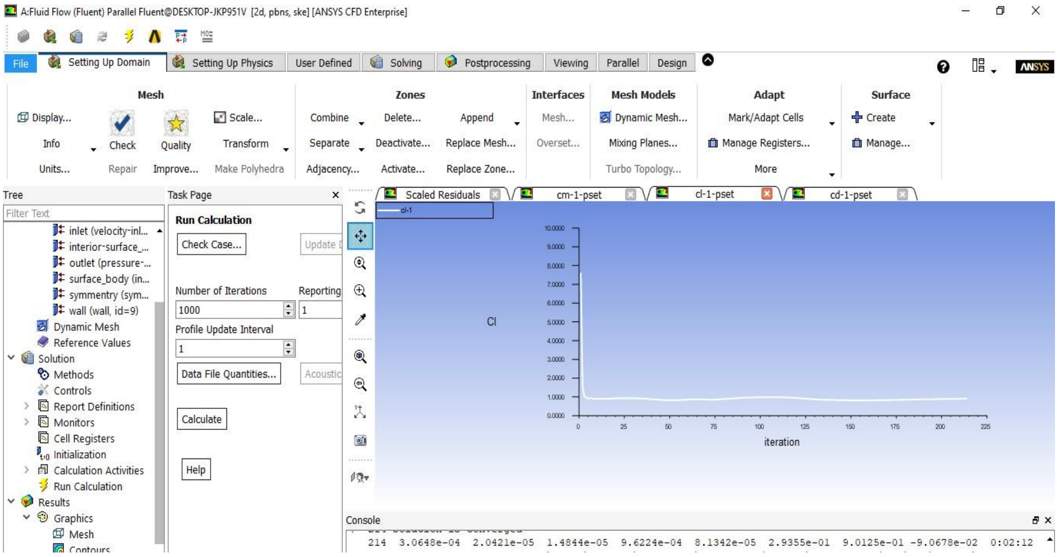


Fig 4.4.6: Lift coefficient plot for NACA 63415

Table 4.4.6: Lift coefficient plot for NACA 63415

AOA	NACA 63415
0	0.29
2	0.41
4	0.52
6	0.62
8	0.72
10	0.79
12	0.85
14	0.91
16	0.92
18	0.90

The lift coefficient plot for NACA 63415 is shown in the above image. In this plot the x axis represents iteration number and y axis represents lift coefficients. At the starting of the processing time, air is hitting the flight and will create turbulence and a curve fluctuation is observed at the beginning of few iterations. After some iteration, it becomes idle and proceeds till the problem is converged. The coefficient of lift reading 0.9 is noted.

CHAPTER - 5

RESULTS AND DISCUSSION

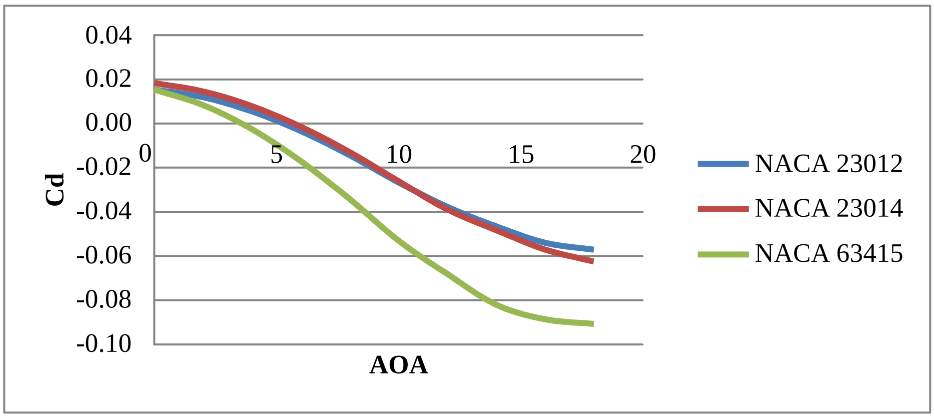
A numerical CFD analysis of NACA profiles is undertaken in this work. ANSYS Fluent software is used to do the analysis. NACA23012, NACA 23014, and NACA 63415 are considered 2D profiles at first. The analysis is done up to 18 AOA to find the optimal aero foil. In tabular columns and graphs, the individual profile drag and lift coefficient are recorded.

5.1 Drag Coefficient

Table 5.1: Drag coefficient of NACA 23012, 23014, 63415

AOA	NACA 23012	NACA 23014	NACA 63415
0	0.02	0.02	0.02
2	0.01	0.01	0.01
4	0.01	0.01	0.00
6	0.00	0.00	-0.02
8	-0.01	-0.01	-0.03
10	-0.03	-0.03	-0.05
12	-0.04	-0.04	-0.07
14	-0.05	-0.05	-0.08
16	-0.05	-0.06	-0.09
18	-0.06	-0.06	-0.09

The table shown in the above is the drag coefficient of AOA for three different profiles. 10 Angle of attack for three different NACA profiles which are NACA 23012; NACA 23014 and NACA 63415 are shown. The angle of attack ranges from 0 to 18.



Graph 5.1: Drag coefficient of NACA 23012, 23014, 63415.

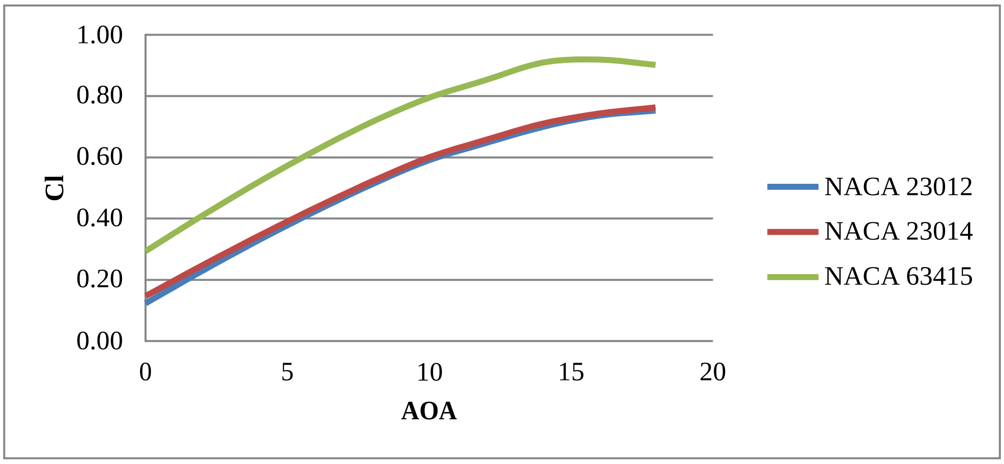
The graphical representation shown above is the drag coefficient for different range of angle of attack for three different NACA profiles. The graph show that NACA 23012 is indicated in blue color, the NACA 23014 is indicated in red color and finally, the NACA 63415 is indicated in green color. The x direction indicates angle of attack and y axis indicates drag coefficient. From the graph, it is seen that the drag coefficient will higher at low angle of attack and low at higher angle of attack. The maximum drag coefficient is same for all profiles which is 0.02. At AOA for 18, the minimum drag coefficient value is -0.06 for NACA 23012 and NACA 23014 and -0.09 for NACA 63415.

5.2 Lift Coefficient

Table 5.2: Lift coefficient of NACA 23012, 23014, 63415

AOA	NACA 23012	NACA 23014	NACA 63415
0	0.12	0.15	0.29
2	0.23	0.25	0.41
4	0.33	0.34	0.52
6	0.42	0.44	0.62
8	0.51	0.52	0.72
10	0.59	0.60	0.79
12	0.65	0.66	0.85
14	0.70	0.71	0.91
16	0.74	0.74	0.92
18	0.75	0.76	0.90

The table shown in the above is the lift coefficient of AOA for three different profiles. 10 Angle of attack for three different NACA profiles which are NACA 23012; NACA 23014 and NACA 63415 are shown. The angle of attack ranges from 0 to 18.



Graph 5.2: Lift coefficient of NACA 23012, 23014, 63415

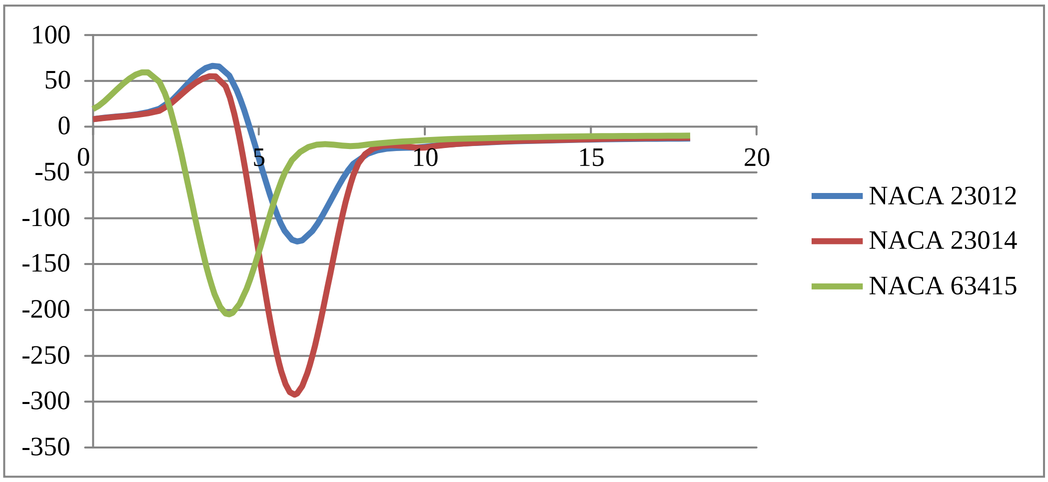
The graphical representation shown above is the drag coefficient for different range of angle of attack for three different NACA profiles. The graph show that NACA 23012 is indicated in blue color, the NACA 23014 is indicated in red color and finally, the NACA 63415 is indicated in green color. The x direction indicates angle of attack and y axis indicates drag coefficient. From the graph, the lift coefficient increases with increase in AOA for all profiles. The maximum lift coefficient is observed in NACA 63415. Minimum lift coefficient is observed in NACA 23012.

5.3 L/d RATIO FOR AERO FOILS

The lift by drag coefficient for different angle of attack for three different profiles is shown in the table 5.3. The three different NACA profiles are NACA 23012, NACA 23014 and NACA 63415 are shown table 5.3.

Table 5.3: L/d ratio of NACA 23012, 23014, 63415

	NACA 23012	NACA 23014	NACA 63415
0	8.04717104	8.046310488	19.02597
2	19.3250654	17.08244074	48.87556
4	60.5619835	43.99621285	-203.997
6	-123.512971	-292.035932	-36.6264
8	-35.6405261	-40.1755196	-20.9754
10	-22.1409285	-22.9681397	-14.9986
12	-17.1322925	-16.8783572	-12.5147
14	-15.0100945	-14.6914398	-11.0869
16	-13.6322563	-13.0152946	-10.3758
18	-13.1640365	-12.2056188	-9.93901



Graph 5.3: L/d ratio of NACA 23012, 23014, 63415

The graphical represent shown above is the l/d ratio with respect to angle of attack for three different NACA profile is shown above. The graph show that NACA 23012 is indicated in blue color, the NACA 23014 is indicated in red color and finally, the NACA 63415 is indicated in green color. The x direction indicates angle of attack and y axis indicates l/d ratio. From the image above, the maximum l/d ratio is seen in NACA 23012 when compared to other and minimum l/d ratio is seen in NACA 23014 when compared to others.

CHAPTER – 6
CONCLUSIONS

The selected profiles 23012, 23014, 63415 are designed in CATIA software and simulated in ANSYS software. The final conclusions made after the above simulation and analysis of the selected profiles are -:

- The form of the airfoil is regarded one of the most essential aspects affecting an aircraft's basic flight, it's critical that the airfoil utilized in its design delivers more lift than drag.
- The lift and drag supplied by the aero foil are significantly reliant on the volume of air that passes through its form, hence the lift and drag coefficients of the aero foil are studied under various NACA configurations.
- The NACA 63415 airfoil offers higher lift than the NACA 23012 and NACA 23014 airfoils, according to the calculations and measurements above. For the airplane employing an airfoil design for its wing to be more efficient, the drag should be reduced.
- The drag produced by the NACA 63415 airfoil is less than that of the NACA 23012 and NACA 23014 airfoils. According to this argument, the NACA 63415 airfoil is superior to the NACA 23012 series airfoil.
- The lift and drag forces over a wing of the NACA type's airfoil shape were estimated via fluent testing.

6.1 Future Scope

- Model analysis can be performed on wings with various profiles to determine the natural frequencies of the wings.
- CFD flow analysis on Airfoil profiles with variable tail sections can be study to meet the practical take off and landing conditions of the planes.

References

[1]. Coulliette, C. and A. Plotkin. “Aerofoil ground effect revisited.” Aeronautical Journal, February 1996, pp. 65-74.

[2] Nuhait, A.O., and M.F. Zedan. “Numerical Simulation of Unsteady Flow Induced by a Flat Plate Moving Near Ground.” Journal of Aircraft, Vol. 30, No. 5, 1993, pp. 611-617.

[3] C. K. Chen and Hsiun, C.H., “Aerodynamic Characteristics of a Two-dimensional Airfoil with Ground Effect.” Journal of Aircraft, Vol. 33, No. 2, 1996, pp. 386-392.

[4] Steinbach, D. “Comment on Aerodynamic Characteristics of a Two-Dimensional Airfoil with Ground Effect.” Journal of Aircraft, Vol. 34, No. 3, 1997, pp. 455-456.

[5] R.H. Chang. And Chun, H.H. “Turbulence Flow Simulation for Wings in Ground Effect with Two Ground Conditions: Fixed and Moving Ground.” International Journal of Maritime Engineering, 2003, pp. 211-228.

[6] J. Leonardi, Barber, T. E., and R.D. Archer. “A Technical Note on the Appropriate CFD Boundary Conditions for the Prediction of Ground Effect aerodynamics.” The Aeronautical Journal, 1999, pp. 545-547.

[7] Saunders, G.H. “Aerodynamic Characteristics of Wings in Ground Proximity.” Canadian Aeronautics and Space Journal, 1965, pp. 185-192.

[8] Tuck, E.O. “A Nonlinear Unsteady One-Dimensional Theory for Wings in Extreme Ground Effect.” Journal of Fluid Mechanics, Vol. 98, Part 1, 1980, pp. 33- 47.

[9] Handbook, F., 2007.AvStop. [Online]Available http://avstop.com/ac/flighttrainghandbook/images4r.jpg [Accessed 4 November 2014].

[10].Scott, 2005. Aerospace Web. [Online] Available http://www.aerospaceweb.org/question/nature/q0237.shtml [Accessed 15 March 2015].