

AERODYNAMICS ANALYSIS OF ELEVONS IN BLENDED WING BODY USING LOW CAMBERED AIRFOIL

Submitted in partial fulfilment of the requirements for the award of

Bachelor of Engineering degree in
Aeronautical Engineering

By

POTHYS M A (Reg. No. 38260032)

HARIHARAN V (Reg. No. 38260015)



DEPARTMENT OF AERONAUTICAL ENGINEERING

SCHOOL OF MECHANICAL ENGINEERING

SATHYABAMA

INSTITUTE OF SCIENCE AND TECHNOLOGY

(DEEMED TO BE UNIVERSITY)

Accredited with Grade "A" by NAAC

JEPPIAAR NAGAR, RAJIV GANDHI SALAI, CHENNAI – 600119

APRIL 2022



SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited with "A" grade by NAAC
Jeppiaar Nagar, Rajiv Gandhi Salai, Chennai - 600 119
www.sathyabama.ac.in



DEPARTMENT OF AERONAUTICAL ENGINEERING

BONAFIDE CERTIFICATE

This is to certify that this Project Report is the Bonafide C:\Users\pothys anand\Downloads\Body of REPORT (2).docx work of POTHYS M A (Reg. No. 38260032) and HARIHARAN V (Reg. No. 38260015) who carried out the project entitled **"DESIGN AND AERODYNAMIC ANALYSIS OF ELEVONS IN BLENDED WING BODY USING LOW CAMBERED AIRFOIL"** under our supervision from September 2021 to April 2022.

Internal Guide

Dr. A. ANDERSON, M.E., Ph.D.,

Head of the Department

Dr. P. BOOMADEVI, ME., PhD.,

Submitted for Viva voce Examination held on _____

Internal Examiner

External Examiner

DECLARATION

We POTHYS M A (Reg. No 38260032) and HARIHARAN V (Reg. No 38260015) hereby declare that the Project Report entitled “**DESIGN AND AERODYNAMIC ANALYSIS OF ELEVONS IN BLENDED WING BODY USING LOW CAMBERED AIRFOIL**” done by us under the guidance of **Dr. A. ANDERSON, M.E., Ph.D.**, is submitted in partial fulfilment of the requirements for the award of Bachelor of Engineering degree in Aeronautical Engineering.

DATE: .04.2021

SIGNATURE OF THE CANDIDATE

PLACE: CHENNAI

ACKNOWLEDGEMENT

I am pleased to acknowledge my sincere thanks to the Board of Management of **SATHYABAMA** for their kind encouragement in doing this project and for completing it successfully. I am grateful to them.

I convey my sincere thanks to **Dr. S. Prakash, M.E., Ph.D.**, Dean, School of Mechanical Engineering and **Dr. P. Booma Devi, M.E., Ph.D.**, Head of the Department, Dept. of Mechanical Engineering for providing me with necessary support and details at the right time during the progressive reviews.

I would like to express my sincere and deep sense of gratitude to my Project Guide **Dr. A. ANDERSON, M.E, Ph.D.**, for his valuable guidance, suggestions, and constant encouragement that paved the way for my project work's successful completion.

I also express my thanks to all Teaching and Non-teaching staff members of the Department of Mechanical Engineering who were helpful in many ways for the completion of the project.

CHAPTER NO:	TITLE		Page no
	LIST OF FIGURES		9
	LIST OF TABLES		10
	ABSTRACT		11
1	INTRODUCTION		12
	1.1	BLENDED WING BODY(BWB)	12
	1.2	ELEVONS	14
	1.3	NACA Airfoil Series	18
2	2.1	LITERATURE REVIEW	20
3		AIM AND SCOPE	23
	3.1	Objective and Scope	23
	3.2	Methodology	23
	3.3	Overview of the Report	24

CHAPTER NO:	TITLE		Page no
4	Design Modelling and Computational Fluid Mechanics		26
	4.1	Design Modelling	26
	4.2	Computational Fluid Mechanics (CFD)	29
5	COMPUTATIONAL DOMAIN AND NUMERICAL SOLUTION		32
	5.1	General	32
	5.2	Meshing	32
	5.3	Boundary Planes	36
	5.4	Numerical Simulation	38
	5.5	Analysis	39

Chapter No	TITLE		Page no
6	RESULTS AND DISCUSSION		45
	6.1	Preliminary Results	45
	6.2	NACA0018	45
	6.3	NACA0024	50
	6.4	Results	56
7	3D DESIGN AND CALCULATION		68
	7.1	3D DESIGN	71
	7.2	CALCULATION	73

CHAPTER NO	TITLE	Page no
8	SUMMARY AND CALCULATION	75

LIST OF FIGURES:

fig.1: Features in the configuration of a blended wing body

fig.2: Blended wing body aircraft

fig.3: Representation of elevons in Northrop series aircraft

fig.4: Elevon function during neutral axis

fig.5: Elevon function during pitch up movement

fig.6: Elevons function during pitch down movement

fig.7: Various functions of elevons

fig.8: a 2D sketch of BWB in different views

fig.9: Design model of BWB in Catia

fig.10: Design model of elevon in Catia

fig.11: Design model of pitching movement in Catia

fig.12: Meshing boundary of the model

fig.13: Network meshing of model

fig.14: Edge meshing of model

fig.15: Face meshing of model

fig.16: Velocity inlet conditions of the model

fig.17: Pressure inlet conditions of the model

fig.18: Boundary condition on the model

fig.19: Pressure path line of NACA 0018

fig.20: Turbulence path line

fig.21: Velocity vector for Reference model

fig.22: Pressure path line for NACA 0024

fig.23: Turbulence path line for NACA 0024

fig.24: Velocity path line for NACA 0024

fig.25: Velocity path line (0, -35) deg

fig.26: Pressure path line (0, -35) deg

fig.27: Velocity path line (4, -35) deg

fig.28: Pressure path line (4, -35) deg

fig.29: Velocity path line (-4, -35) deg

fig.30: Pressure path line (-4, -35)

fig.31: Velocity path line (8, -35) deg

fig.32: Pressure path line (8, -35) deg

LIST OF TABLES:

6.3.3) Tabulation of aerodynamic efficiency of two airfoils

6.3.4) Comparison of the values for the NACA0018 with different angles of attack:

ABBREVIATION AND FORMULAS:

NACA- NATIONAL ADVISORY COMMITTEE FOR AERONAUTICS

AOA– ANGLE OF ATTACK

BWB–BLENDED WING BODY

NASA– NATIONAL AERONAUTICS AND SPACE ADMINISTRATION

CATIA– COMPUTER-AIDED THREE-DIMENSIONAL INTERACTIVE APPLICATION

CFD - COMPUTATIONAL FLUID DYNAMICS

ANSYS- ANALYSIS OF SYSTEMS

PLA - POLYLACTIC ACID

ABSTRACT: Some technology research and development attempts exist to combine the functions of aircraft flight control systems such as ailerons, elevators, elevons and flaps into wings to perform the aerodynamic purpose with the advantages of less mass, cost, drag, inertia, complexity, and radar cross-section for stealth. However, the main disadvantage is that when the elevons move up in a single direction to raise the pitch of the aircraft, generating secondary lift. the Elevons are aircraft control surfaces that integrate the functions of the elevator (used for pitch control) and the aileron (used for roll control), hence the name. They are frequently used on tailless aircraft such as flying wings. To avoid additional lift, to reduce the cambered length of the airfoil at elevon spot or downward curvature re of wing which is flexible for that we are using two low cambered airfoils namely NACA0018 and NACA0024. By using the NACA00018, elevon reduces the maximum lift produced during pitch up and results in greater aerodynamic efficiency and performance of the wing. Boeing X-48 b is a blended wing body aircraft which find beneficial in reducing sizing of aircraft, where it reduces drag and increases fuel efficiency when compared to conventional aircraft.

INTRODUCTION

1.1) BLENDED WING BODY: The Blended Wing Body (BWB) configuration is the future of aviation that can offer very high fuel economy along with a large payload and quieter & cleaner operation. It is a hybrid shape, with fuselage and wings blended smoothly to make a single lifting surface. The BWB resembles a flying wing but also incorporates features from conventional aircraft. This combination gives several advantages over conventional tube and wing airframes. As the airframe encompasses an aerofoils shape body blended with the high lift wings, it allows the whole airframe to generate lift that improves the fuel economy. It is expected that BWB aircraft would improve lift to drag ratio by 50 % and fuel efficiency by 20 to 25 % in comparison to the conventional configuration. The blended wing body (BWB), also known as the blended body or hybrid wing body (HWB), is a fixed-wing aircraft having no clear dividing line between the wings and the main body of the craft. The aircraft has distinct wing and body structures, which are smoothly blended with no clear dividing line. This contrasts with a flying wing, which has no distinct fuselage, and a lifting body, which has no distinct wing. A BWB design may or may not be tailless. As design work at Douglas progressed, new benefits of the BWB were discovered. In almost every case the findings were favourable for the BWB relative to a Tube-and-Wing.

The findings and summary of the Blended wing body are:

- The airframe shields the community from most of the engine noise
- Supersonic flow above the cabin prevents forward-radiated noise from entering the cabin
- The nacelles are protected from ground handling equipment
- Intake and exhaust hazards are eliminated for the ground personnel and equipment
- Thrust reversers do not blank the stabilizing surfaces, a major problem for today's transports on icy runways
- The fuel tanks are protected from rotor-burst

- The pressure vessel and passengers are protected from rotor-burst
- The primary wing structure is largely protected from rotor-burst - Wingtip-stall is much improved since the elevons reduce air load near stall
- Engine inlets are protected from the stall at a high angle of attack since the airframe directs the flow to the inlet
- The majority of the centre-body is stall-free up to and beyond the wing stall.
- The mid-span planform kink is stalling critical to protect the wingtips from the stall and the engines from distortion
- Passenger cabin loading, unloading, and emergency egress are faster due to multiple short aisles
- Ditching stability and integrity are improved by the large belly surface
- The traditional tails and associated systems are eliminated.
- The entire high lift flap system is removed
- No control surfaces are behind the wing where they are vulnerable to stalled wakes
- The wing shape and lack of tails reduce the size of the ice protection system

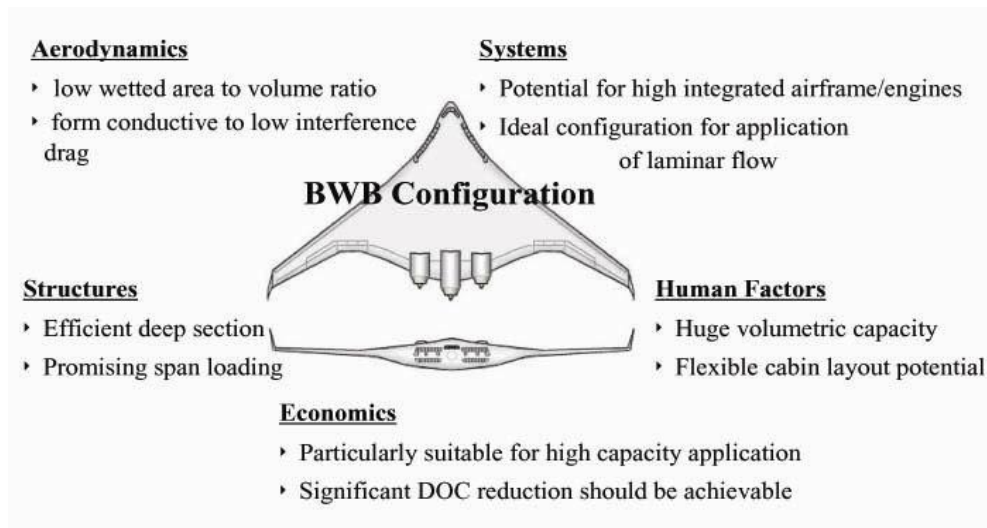


fig.1: Features in the configuration of blended wing body



Fig.2: Blended wing body aircraft

1.2) ELEVONS: Elevator + aileron = Elevon

The combined function of elevator and aileron is elevon. Elevons are aircraft control surfaces that combine the functions of the elevator (used for pitch control) and the aileron (used for roll control). They are frequently used on tailless aircraft such as flying wings. An elevon that is not part of the main wing, but instead is a separate tail surface, is a stabilator (but stabilators are also used for pitch control only, with no roll function, as on the Piper Cherokee series of aircraft). The word "elevon" is a portmanteau of elevator and aileron. Elevons are installed on each side of the aircraft at the trailing edge of the wing. When moved in the same direction (up or down) they will cause a pitching force (nose up or nose down) to be applied to the airframe. When moved differentially, (one up, one down) they will cause a rolling force to be applied. These forces may be applied simultaneously by appropriate positioning of the elevons, for example, one wing's elevons completely down and the other wing's elevons partly down. The elevon is supported by an aircraft frame structure, for pivotal movement about a hinge axis. The surface is deployed against an aerodynamic load which is a function of surface deflection and imposes a torque on the flight control surface wanting to rotate it back to a neutral trim position. A counterbalancing hydraulic actuator is connected between the frame structure and the flight control surface. This actuator is oriented to oppose a counterbalancing torque on the flight control surface acting in opposition to the torque imposed by the

aerodynamic load. A controllable, separate actuator is interconnected between the frame structure and the flight control surface and is operated for positioning the flight control surface. Elevons of modern pure delta wing aircraft can deflect 25deg to 30deg throughout the operation. At normal operation, no loads are acting on the control surface.



Fig.3: Representation of elevons in Northrop series aircraft

1.2.1) ELEVATORS: Elevators are responsible for the lift and pitching movement of aircraft

The functions of elevators are:

- The elevators respond to a forward or aft movement of the control column or control stick.
- When the pilot moves the controls forward, the elevator surface is deflected downwards.
- This increases the camber of the horizontal stabilizer increasing lift.
- The additional lift on the tail surface causes rotation around the lateral axis of the aircraft and results in a nose-down change in aircraft attitude.

- The opposite occurs with an after movement of the flight deck controls.

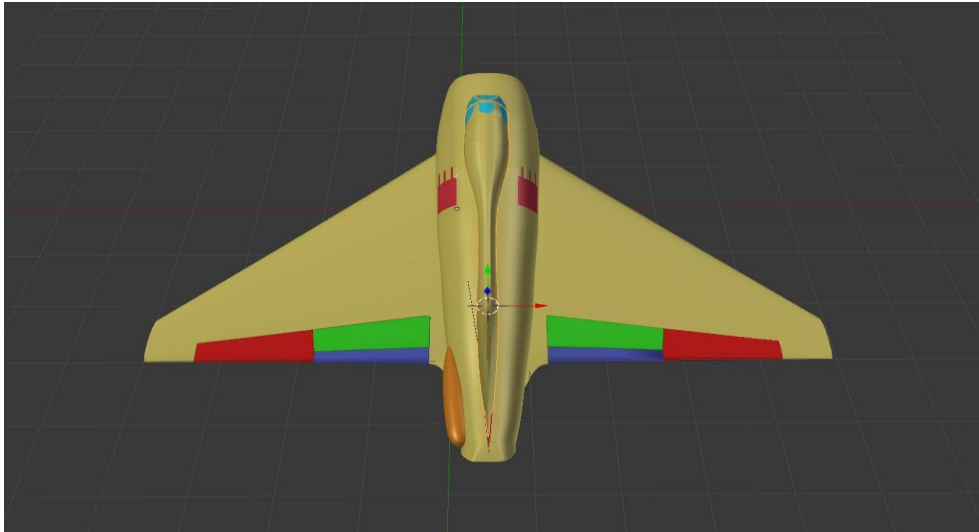


fig.4: Elevon function during neutral axis



fig.5: Elevon function during pitch up movement



fig.6: Elevons function during pitch down movement

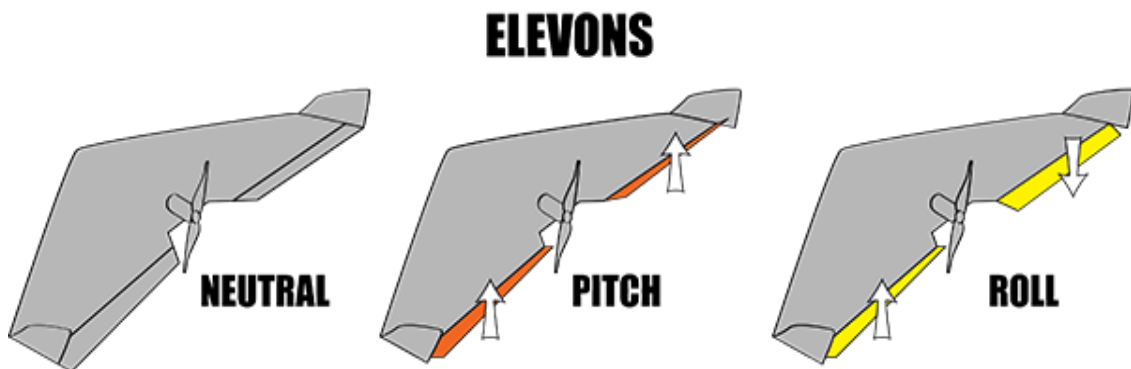


fig.7: Various functions of elevons

1.2.2) AILERON: An aileron is a hinged flight control surface usually forming part of the trailing edge of each wing of a fixed-wing aircraft. Ailerons are used in pairs to control the aircraft in roll or combine the elevator for the aircraft's pitching trim, which normally results in a change in flight path due to the tilting of the lift vector. Ailerons are quite often situated near the wing tip but may sometimes also be situated near the wing root. Modern large transport aircraft (e.g., bomber Boeing B-2) also have a second pair of ailerons on their wings, and the terms “outboard aileron” and “inboard aileron” are used to describe these positions, respectively.

With the development of the control surface system, several combination types of ailerons occur. Combining an aileron and a flap is called a flaperon, which has the functions of both ailerons and flaps. The ailerons are combined with the elevators to form an elevon, which is applied to delta-winged aircraft. Some high-speed aircraft are equipped with inner ailerons to reduce the torsional deformation of the wings caused by the deflection of the ailerons.

1.3) NACA Airfoil series: The early NACA airfoil series, the 4-digit, 5-digit, and modified 4-/5-digit, were generated using analytical equations that describe the camber (curvature) of the mean-line (geometric centerline) of the airfoil section as well as the section's thickness distribution along the length of the airfoil. Later families, including the 6-Series, are more complicated shapes derived using theoretical rather than geometrical methods. Before the National Advisory Committee for Aeronautics (NACA) developed these series, airfoil design was rather arbitrary with nothing to guide the designer except experience with known shapes and experimentation with modifications to those shapes. This methodology began to change in the early 1930s with the publishing of a NACA report entitled *The Characteristics of 78 Related Airfoil Sections from Tests in the Variable Density Wind Tunnel*. In this landmark report, the authors noted that many similarities between the airfoils were most successful, and the two primary variables that affect those shapes are the slope of the airfoil mean camber line and the thickness distribution above and below this line. They then presented a series of equations incorporating these two variables that could be used to generate an entire family of related airfoil shapes. As airfoil design became more sophisticated, this basic approach was modified to include additional variables, but these two basic geometrical values remained at the heart of all NACA airfoil series. The NACA four-digit wing sections define the profile by the First digit describing maximum camber as a percentage of the chord. The second digit describes the distance of maximum camber from the airfoil leading edge in tenths of the chord. The last two digits describe the maximum thickness of the airfoil as a percent of the chord

1.3.1) NACA0018:

The first digit (0)-maximum camber as a percentage of the chord or maximum camber is 0% of 2d airfoil chord length

The second digit (0)-The distance of maximum camber from the airfoil leading edge in the tenths of the chord or location of maximum camber is at 0/10th (or 0%) of the chord line, from the L.E

Third and fourth digit (18)-The maximum thickness of the airfoil as a percent of chord or the maximum thickness is 18% of C

CHAPTER 2

LITERATURE REVIEW

Numerous researches and studies of the wind lens technology of wind turbines were done all around the world. Many successful results have been arrived at in the history of the research. A few research papers deal with shrouded wind turbines and some of these are highlighted below.

2.1) FLIGHT CHARACTERISTICS AT LOW SPEED OF DELTA WING MODELS

Marion O. McKinney, Jr., and Hubert M. Drake

The flight characteristics of the models with 76° or 83° sweep back or aspect ratios of 1 or less were unsatisfactory because of unstable rolling oscillations at high lift coefficients or because of excessive changes in static longitudinal stability over the lift range. An investigation has been made in the Langley free-flight tunnel, therefore, to study the flying characteristics of some models with low- aspect-ratio delta wings. This investigation was exploratory and was intended only to provide a preliminary survey of the flying characteristics of delta wings over a range of sweep angles to determine whether a detailed study of delta wings is justified.

DESIGN VARIABLES: Aerodynamic characteristics, Airfoil section, Delta-wing models flying in the Langley free-flight tunnel.

2.2) DESIGN OF THE ACTIVE ELEVON ROTOR FOR LOW VIBRATION

Mark V. Fulton Research Scientist

The present study is focused on elevon effectiveness, that is, the Elevon's ability to reduce all six components of the non-rotating 4/rev hub loads. Some design parameters have been kept fixed in this study, while others have been varied to determine their influence on elevon effectiveness. The US Army Aero flight dynamics Directorate has undertaken a new research program called the Active Elevon Rotor (AER) Focus Demo. This program includes the design, fabrication,

and wind tunnel testing of a four-bladed, 12.96 ft. diameter rotor with one or two on-blade elevons per blade.

DESIGN VARIABLES: The use of six elevon locations is a simplification which will likely be abandoned during future work – a design goal for the AER is that only one or two independently controlled elevons be used on each blade.

2.3) DESIGN AND ANALYSIS OF ELEVON ACTUATING LINKAGE MECHANISM AT ULTIMATE LOADING CONDITION

S. Saranya¹, B. Malarvizhi¹, T. Balamurugan¹, N. Tamil Selvam

The movement of Elevon assemblies is controlled by the Elevon actuator, which is connected between Elevon and Wing Structure. The required movement of Elevon is controlled by Digital Flight Control Computer (DFCC). This project deals with the design modification of Connecting Horn and Connecting Rib linkage mechanism of inboard elevon of pure delta aircraft to counterbalance the Aerodynamic and flutter loads during flight.

DESIGN VARIABLES: Whenever a bolt is designed it is taken care that the hole in which a bolt is to be inserted is always greater than the bolt diameter. To minimize the error due to the gap produced when the bolt is inserted into a hole, a fitting factor is introduced.

2.4) DESIGN OF ELEVONS, WINGS, AND PERFORMANCE INVESTIGATION FOR A BLENDED WING BODY UAV

Ashutosh Kumar, Raghavendra Gautam

The Design phase is highly iterative, through this research paper, an attempt has been made to develop a methodology for the selection and investigation of control surfaces against requirements that make BWB UAVs more helpful for practical use

and increase the lift and endurance efficiency of the hybrid VTOL- Blended wing body aircraft. The design process of such UAVs which are used in HADR missions (Humanitarian Aid and Disaster Relief) is not well understood since enough literature is not available and with complex systems and variables present, the design and its optimization of a UAV is a serious challenge. Hence this paper on control surfaces optimization and the effect of winglets on the performance of UAVs is our attempt to provide future researchers and engineers in designing.

DESIGN VARIABLES: For the BWB wing, 1. Elevon chord/span (C_e/b_e), 2. Planform area of elevon (S_e), 3. Elevon max deflection, 4. Location of elevon inboard

CHAPTER 3

AIM AND SCOPE

3.1) Objective and scope:

3.1.1) Aim: The objective of the current study is to avoid secondary lift produced by the aircraft (BWB) while elevon moves up in unison direction by reducing the camber length of an airfoil at elevons or downward curvature of the wing (flexible)

3.1.2) Scope: The scope of this study discusses the analysis variation of the aerodynamic efficiency of two different low-cambered elevons. The two low camber airfoils that determine greater efficiency will undergo the analysis variation in different angles of attack with constant elevon deflection, further establishing the 3d CFD model of BWB with varying AOA as follows

Aoa:0°and elevon deflection: -35°: When BWB aircraft X-48 b is subjected to an angle of attack of 0°and elevon deflection is -35° where (-) is referred to as counterclockwise direction

Aoa:4°and elevon deflection: -35°: When BWB aircraft X-48 b is subjected to the angle of attack of 4°and elevon deflection is -35° where (-) is referred to as counterclockwise direction

AOA: -4°and elevon deflection: -35°: When BWB aircraft X-48 b is subjected to the angle of attack of -4°and elevon deflection is -35° where (-) is referred as counterclockwise direction

Aoa:8°and elevon deflection: -35°: When BWB aircraft X-48 b is subjected to the angle of attack of 8°and elevon deflection is -35° where (-) is referred to as counterclockwise direction

3.2) Methodology:

1) Literature survey

- 2) Taking a Boeing-X48 Blended wing body and Selection of 2 NACA Low Cambered Airfoils (NACA 0024, NACA 0018) for elevons
- 3) Creating a 3d model of Boeing - X48 Elevons by using these NACA Airfoils in CATIA software.
- 4) Convert the 3D Model into STEP files for analysis
- 5) Import the STEP files into ANSYS Fluent in ANSYS Workbench software
- 6) Creation of 3D boundary, Meshing, Computational domain
- 7) CFD analysis on NACA airfoil (NACA 0024, NACA 0018)
- 8) CFD analysis on other 2 NACA airfoils (NACA 0024, NACA 0018)
- 9) Conclusion and discussion (Lift, Drag, Velocity, Pressure, C_l , C_d)
- 10) Selection and Fabrication of better results among 2 NACA airfoils by comparison of results
- 11) wind tunnel Experimental analysis on BWB with elevons
- 12) comparing numerical simulation and experimental results

3.3) Overview of this Report:

A full-scale study of flow around the Blended wing body and low cambered elevon is done using 3D CFD simulation. The aims and objectives are given in Chapter 1. In Chapter 2, the literature studies are compiled. In Chapter 3, the theories of

computational fluid dynamics are explained. 3D Modeling of BWB with modification is described in Chapter 4. The meshing of the BWB airfoil using Ansys Fluent software and three-dimensional simulation procedures using fluent are given in Chapter 4. The velocity and aerodynamic forces are changed and efficient modification is considered and the results are given in Chapter 5. The 3D design for the wind-tunnel experiments and calculations are presented in Chapter 7. The conclusions are summarized in chapter 8.

CHAPTER 4

DESIGN AND COMPUTATIONAL FLUID DYNAMICS

4.1) DESIGN MODELLING: The diagrammatic sketch of airfoil NACA0018 and NACA0024 were taken from the airfoil tools website, giving the coordinates of the airfoil. For reference design purposes, the aircraft is a combination of fuselage and wing as a single body and the trailing edge of the aircraft consists of elevons.

NACA 0018 airfoil coordinates are created in CATIA by using view macros in Microsoft excel according to the geometry of BWB(X-48B) generated to complete the model.

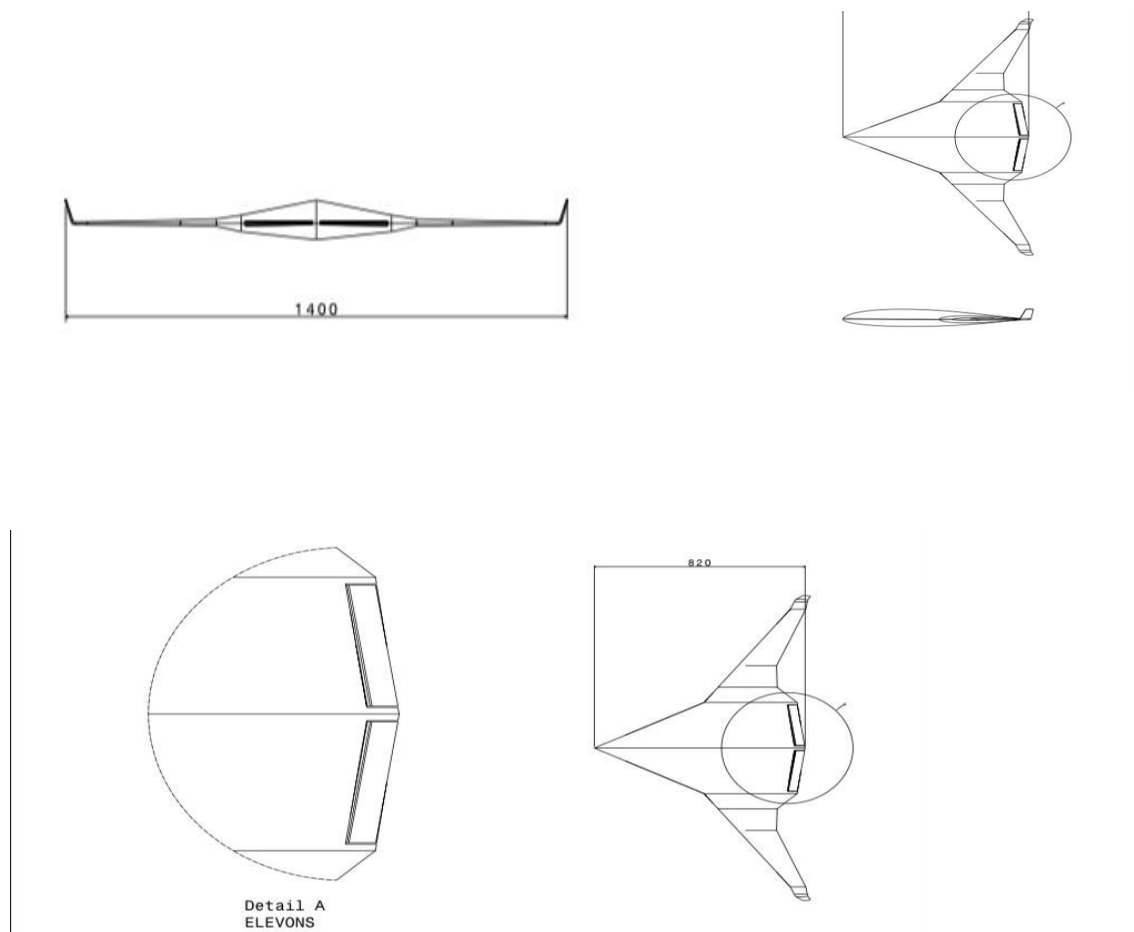


fig.8: a 2D sketch of BWB in different views

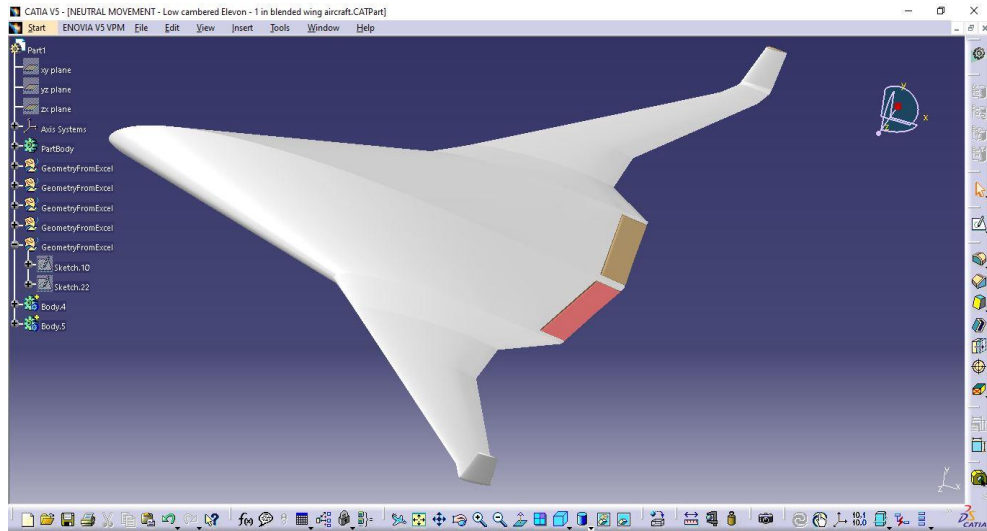


fig.9: Design model of BWB in Catia

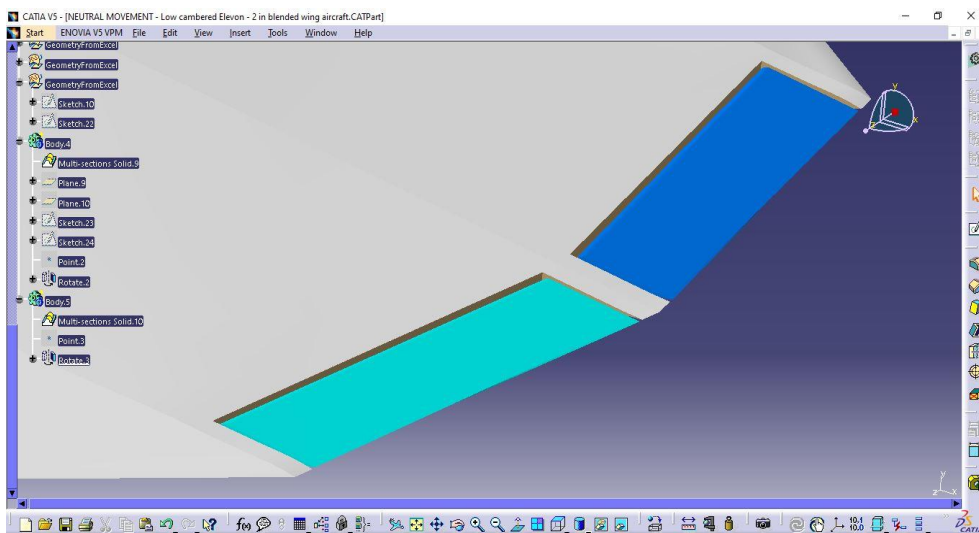


fig.10: Design model of elevon in Catia

The following factors are used to design the Blended Wing Body aircraft:

Airfoil used: NACA 0018 and NACA 0024

Front view span:1400 m

Top view span:820 m

4.1.1) Requirements:

Software used: CATIA V5

Processor: Intel Core i3

RAM: 4GB

Operating System: Windows10

4.1.2) Features Used:

EXTRUDE: This feature is used to extrude the length of the airfoil of the aircraft

ROTATE: This option is used to vary the angles of attack of aircraft and as well as elevon from -35 to 35, for our purposes we are changing the AOA of aircraft (-4,0,4,8)

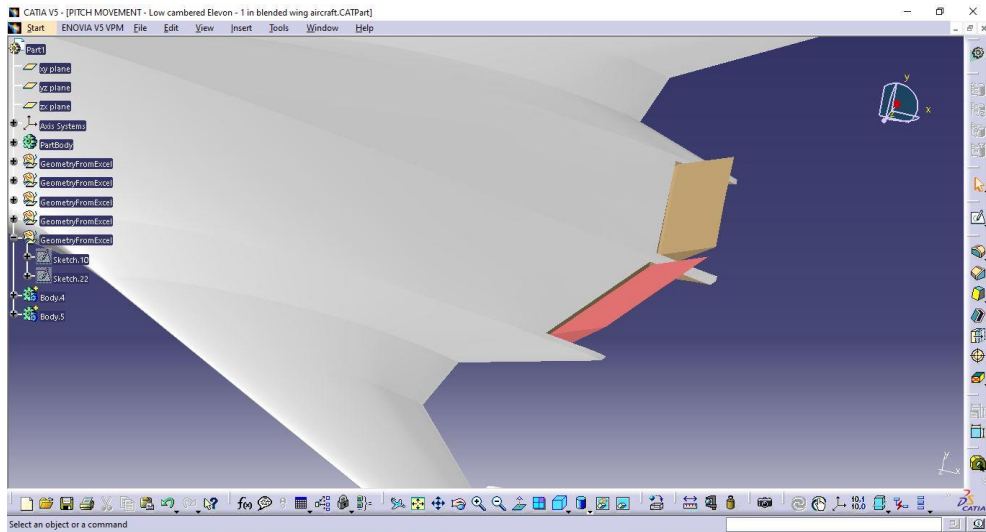


fig.11: Design model of pitching movement in Catia

4.2) COMPUTATIONAL FLUID DYNAMICS

4.2.1) GENERAL

With the rapid advances in computing power, the application of Computational Fluid Dynamics (CFD) has become practiced for various wind flow-related studies. Even though Direct Numerical Simulation (DNS) is limited to supercomputer users for studying and generating benchmark solutions for low Reynolds numbers of up to a few thousand. Various turbulence models like Large Eddy Simulation (LES) and Reynolds Averaged Navier-Stokes Simulation (RANS) turbulence models are being used for numerically simulating turbulent flow around bluff bodies. At high Reynolds numbers, time-varying flow quantity like velocity and pressure consists of a time-averaged mean component, a periodic component and an abandoned turbulent fluctuating component. The periodic component is usually due to the vortex shedding in the wake of a bluff body.

LES which is based on spatial averaging resolves not only the periodic component but also random turbulent fluctuating component with length scales more than the cell size while modelling the turbulence with length scales less than the cell size. However, Unsteady RANS(URANS) which is based on ensemble averaging resolves only the periodic component while modelling the turbulence component. Even though LES predictions are more accurate than RANS predictions, RANS turbulence models especially two-equation turbulence models like $k-\epsilon$ have been very popular because they require less computer memory and CPU time than LES.

4.2.2) Continuity equation: The continuity equation describes the transport of some quantities like fluid or gas. For example, the equation explains how a fluid conserves mass in its motion. Many physical phenomena like energy, mass, momentum, natural quantities, and electric charge are conserved using the continuity equations. The continuity equation provides very useful information about the flow of fluids and their behavior in a pipe or hose. Continuity Equation is applied on tubes, pipes, rivers, ducts with flowing fluids or gases and many more.

The differential form of the continuity equation is:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0$$

Where,

t

= Time

ρ

= density

u

= flow velocity vector field.

4.2.3) Turbulence models: The turbulent flows are characterized by fluctuating velocity fields. These fluctuations mixed transported quantities to fluctuate such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Since these fluctuations can be small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the instantaneous (exact) can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms of known quantities.

FLUENT provides the following choices of turbulence models:

- Spalart - Allmaras model
- K- ϵ models
 - Standard k- ϵ model
 - Renormalization-group (RNG) k- ϵ model

- Realizable k - ϵ model
- k - ω models
 - Standard k - ϵ model
 - Shear-stress transport (SST) k - ω model
 - ν^2 - f model (addon)
- Reynolds stress model (RSM)
 - Linear pressure-strain RSM model
 - Quadratic pressure-strain RSM model
 - Low-Re stress- ω RSM model

CHAPTER 5

COMPUTATIONAL DOMAIN AND NUMERICAL SIMULATION

5.1) General:

The pre-processing of model development and meshing of the model has been carried out using GAMBIT software. GAMBIT is designed to help analysts and designers build meshing models for use in Computational Fluid Dynamics (CFD) and other scientific applications. GAMBIT receives user input utilizing its Graphical User Interface (GUI).

In this study, the various parts of the model include the shroud, diffuser body, and the brim which are modelled by using this software. The profiles are generated with the help of coordinates available which have been generated. Meshing is done after the modelling processes.

5.2) Meshing:

Meshing is the process of dividing a continuous unit into smaller individual units.

5.2.1) Edge Mesh:

The domain and the 2D model surface are discretized using the edge mesh tool in the gambit. This is done to break the continuous domain into smaller units called Grids.

The edge mesh for the domain was done with the following features:

Type: Last Length

Last Length measure: 0.025m

Spacing: 0.2m

The edge mesh for the model was done with the following features;

Type: Successive Ratio

Ratio: 1

Spacing: 0.029m

5.2.2) Face Mesh:

For the carrying out face mesh, faces were created by splitting the entire domain into individual parts. Then, the faces were later joined as the meshing for each face match. Mesh far away from the model was made coarse and near the model was done finer. This will help in reducing the running time of the flow analysis.

Face mesh for the domain was done with the following features.

Type: Map

Element: Quadrilateral

Spacing: 0.2m

Face mesh near the model was done with the following features:

Type: Map

Element: Quadrilateral

Spacing: 0.027m

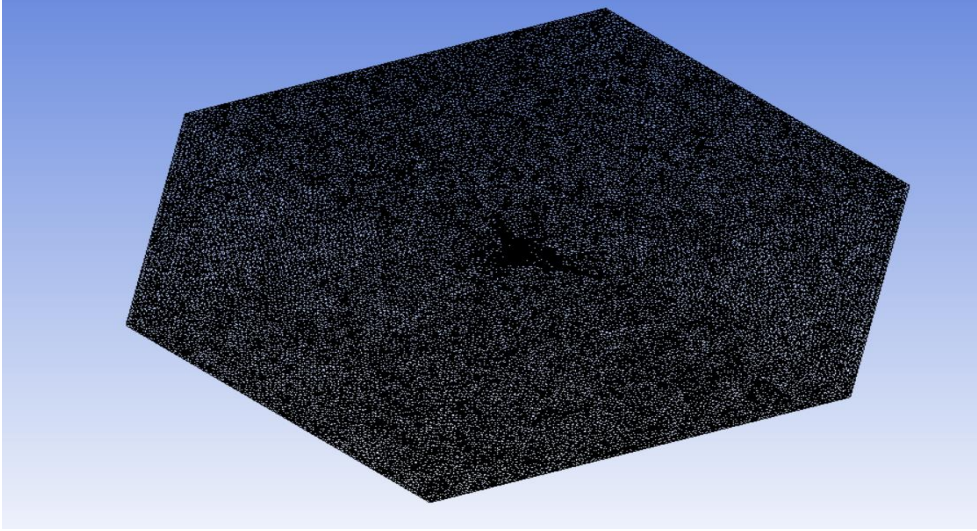


fig.12: Meshing boundary of the model

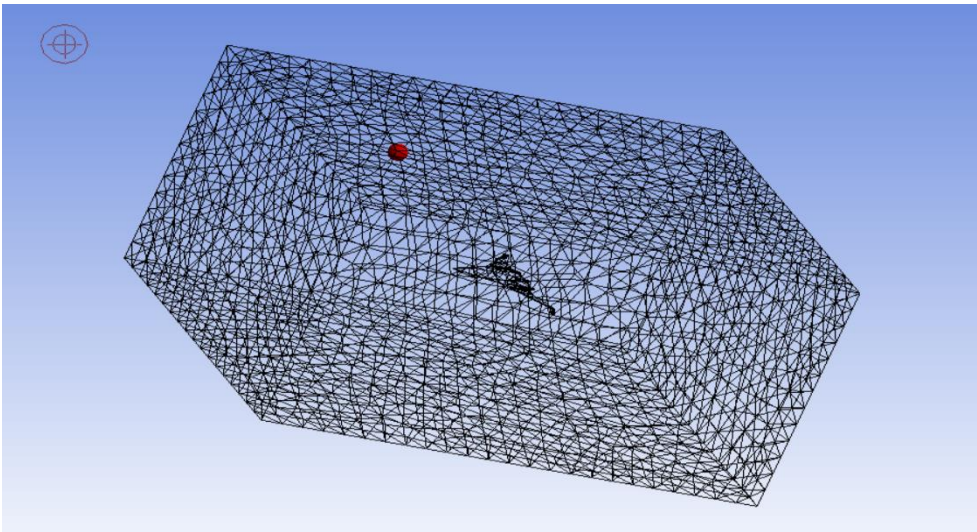


fig.13: Network meshing of model

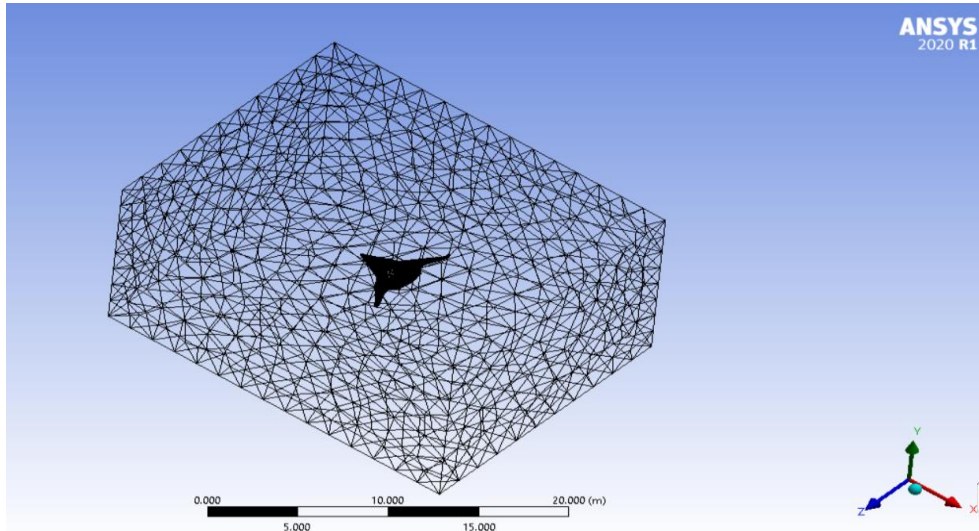


fig.14: Edge meshing of model

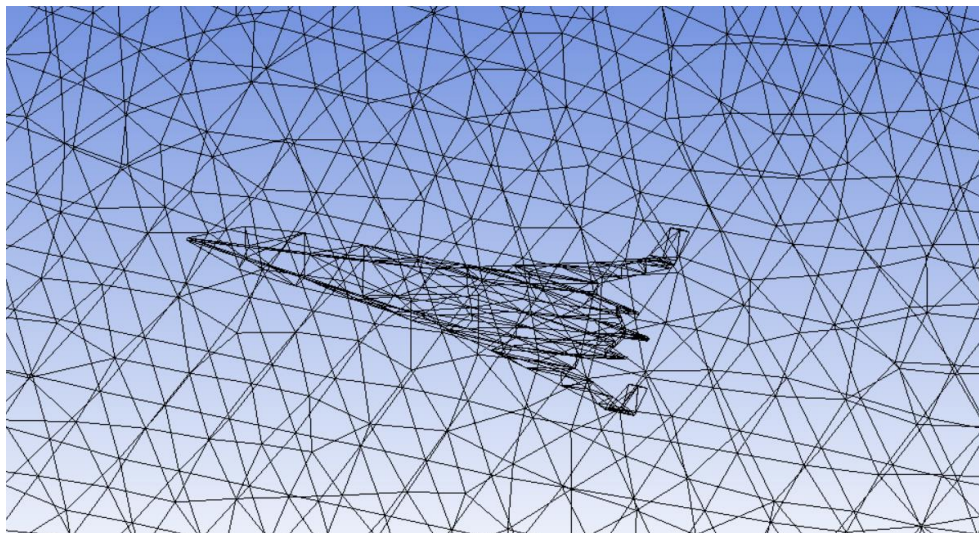


fig.15: Face meshing of model

5.3) Boundary Plane:

The boundary plane set keeps in mind the axis-symmetric feature. The five boundary conditions set in this modelling are as follows:

- Velocity Inlet Boundary Conditions
- Pressure Outlet Boundary Conditions
- Wall Boundary Conditions
- Symmetry Boundary Conditions
- Axis Boundary Conditions

5.3.1) Velocity Inlet Boundary Conditions:

The domain inlet wall is set as velocity inlet boundary. Velocity inlet boundary conditions are used to define the flow velocity, along with the relevant scalar properties of the flow inlets. The total (or Stagnation) properties of the flow are not fixed, so they will rise to whatever value is necessary to provide the prescribed velocity distribution.

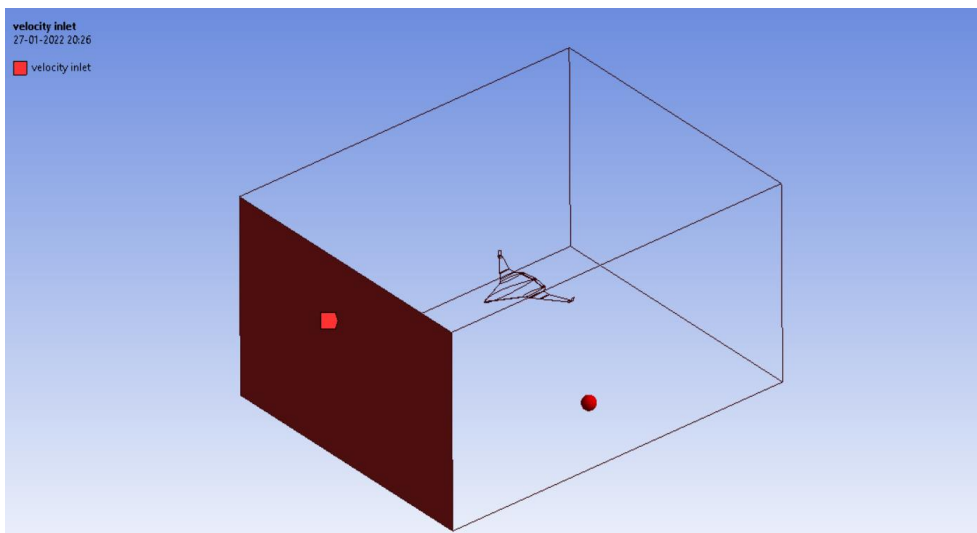


fig.16: Velocity inlet conditions of the model

5.3.2) Pressure Outlet Boundary Conditions:

The domain outlet wall is set as a Boundary Pressure outlet boundary. Pressure outlet boundary conditions require the specification of a static(gauge) pressure at the outlet boundary. The value of static pressure specified is used only while the flow is subsonic.

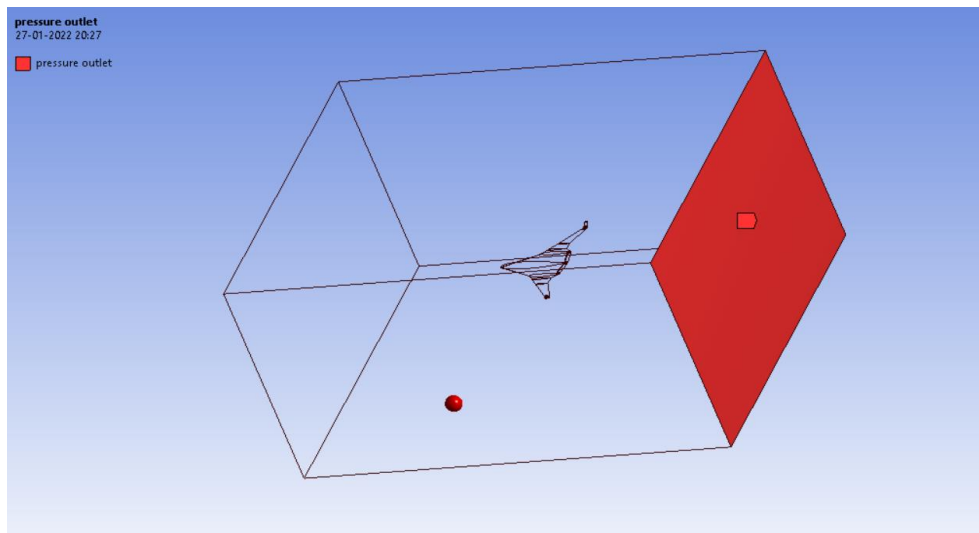


fig.17: Pressure inlet conditions of the model

5.3.3) Wall Boundary Conditions:

The model wall is set as a wall boundary. Wall boundary conditions are used to bound fluid and solid regions. In viscous flows, the no-slip boundary condition is enforced at walls by default, but it can be specified that a tangential velocity component in terms of translational or rotational motion of the wall boundary, or model a slip wall by specifying shear. The shear stress and the heat transfer between the fluid and wall are computed based on the flow details in the local flow field.

5.3.4) Axis Boundary Conditions:

The center axis of the axis-symmetric model is set as the axis boundary. The axis boundary type must be used as the centerline of an axis-symmetric geometry. It can also be used for the centerline of a cylindrical-polar quadrilateral or hexahedral grid. No boundary conditions are required at axis boundaries.

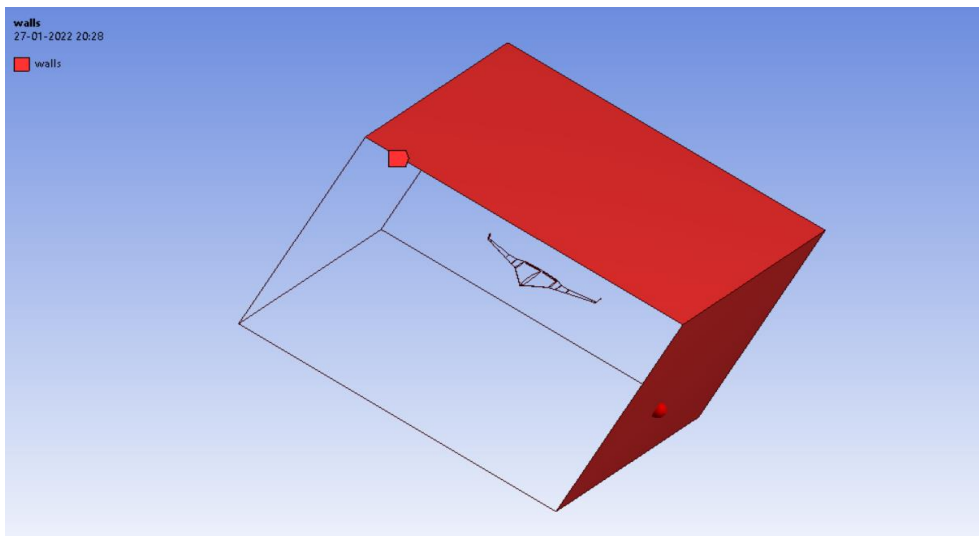


fig.18: Boundary condition on the model

5.4) NUMERICAL SIMULATION USING FLUENT 6.3:

5.4.1) OVERVIEW OF FLUENT:

Numerical simulation has been carried out using fluent software. This software is used for modelling fluid flow and heat transfer in complex geometries. It provides complete mesh flexibility, including the ability to solve the flow problems using unstructured meshes that can be generated about complex geometries with the relative ease. It is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true

dynamic memory allocation, efficient data structures, and flexible solver control are all possible.

All functions required to compute a solution and display the results are accessible in FLUENT through an interactive, menu-driven interface. They reference procedural steps for solving a problem in FLUENT including:

- 1) Define the modelling goals
- 2) Create the model geometry and grid
- 3) Set up the solver and physical models
- 4) Compute and monitor the solution
- 5) Examine and save the results
- 6) Consider revisions to the numerical or physical model parameters, if necessary

5.5) ANALYSIS:

The analysis has been carried out in fluent by importing the meshed file saved in gambit. The steps that followed are given below which include all the conditions and the boundary values for the problem statement.

Grid Check

```
Domain Extents:
  x-coordinate: min (m) = 0.000000e+000, max (m) = 2.529700e+001
  y-coordinate: min (m) = 0.000000e+000, max (m) = 9.761000e+000
Volume statistics:
  minimum volume (m3): 2.198033e-004
  maximum volume (m3): 4.570068e-001
  total volume (m3): 2.468763e+002
Face area statistics:
  minimum face area (m2): 1.891260e-002
  maximum face area (m2): 6.842351e-001
Checking number of nodes per cell.
Checking number of faces per cell.
Checking thread pointers.
Checking number of cells per face.
Checking face cells.
Checking bridge faces.
Checking right-handed cells.
Checking face handedness.
Checking face node order.
Checking element type consistency.
Checking boundary types:
Checking face pairs.
Checking periodic boundaries.
Checking node count.
Checking nosolve cell count.
Checking nosolve face count.
Checking face children.
Checking cell children.
Checking storage.
Done.
```

Fig19. Grid check-in Fluent software

5.6) Solver and Material Selection and Operating Condition Defining:

Solver:

The solver is defined first. The energy equation is taken into consideration. The viscous medium is also taken. First, the analysis is carried out using turbulent flow with the k- ϵ model, specifically realizable.

The following conditions are taken in the solver:

Table 2. Solver specification:

Solver	Pressure Based
Formulation	Implicit
Time	Unsteady
Space	Axisymmetric
Velocity Formulation	Absolute
Porous Formulation	Physical Velocity
Gradient options	Green-Gauss Cell
Viscous Model	k-ε

Selection of Material and Operating Conditions:

The material selected is air. The Properties of the fluid are as follows.

The following values are entered in Viscous and Material Selection.

Table 3. Material Properties

Material Type	Fluid
Fluid	Air
Density	1.225 kg/m ³
Viscosity	1.7894 x 10 ⁻⁵ kg/ms
Pressure	101325 Pa

5.7) Boundary conditions:

The boundary conditions selected are as follows:

- Velocity Inlet Boundary Conditions for the domain inlet.
- Pressure Outlet Boundary Conditions for the domain outlet.

- Wall Boundary Conditions for the model wall.
- Symmetry Boundary Conditions for the upper edge of the domain.
- Axis Boundary Conditions for the bottom edge of the domain.

The velocity inlet conditions set are as tabulated below.

Table 4. Velocity inlet conditions & Pressure Outlet conditions:

Inlet	Velocity Inlet, Velocity =50 m/s
Specific method	Intensity & Viscosity ratio
Turbulent Intensity	0.1
Viscosity ratio	10
Outlet	Pressure Outlet
Specific method	Intensity & Viscosity ratio
Turbulent Intensity	0.1
Viscosity ratio	10

5.8) Residual Monitoring:

The computation is done from the inlet.

Residual Monitoring is done and convergence criteria are set up. The convergence criteria of various parameters are listed below.

Continuity - 0.000001

X-Velocity - 0.000001

Y-Velocity - 0.000001

k-ε - 0.000001

Later the reference values are entered and the number of iterations is then set up and iterations start. The iteration continues till the convergence is reached.

Report Reference Values:

In reporting parameters, the reference values are given in the following

5.9) Report Reference Values

In reporting parameters, the reference values are given as follows

Table 5. Reference Values

Density kg/m ³	1.225
Enthalpy, J/kg	0
Length, m	1
Pressure, Pa	0
Temperature, K	288.16
Velocity, m/s	4
Viscosity, kg/m s	0.17894×10^{-4}
The ratio of specific heat	1.4
Reference Zone	Fluid

In the reference values, the area of the model differs in each case of dimension changes.

5.10) Iteration:

The iteration process is started after setting the reference values. The following values are given as the requisites in the iteration.

Table 6. Iteration Values

Time Step size	0.001
No. of Time Steps	1000
Reporting Interval	10
Maximum Iteration per time step	20

Based on all the above values tabulated, the analysis of the flow was done in fluent. The result can be seen as the plot in the fluent software.

Even though unsteady simulations are carried out, the solution is converged to a steady-state solution due to the simulations correspond to the 2-D Axisymmetric condition.

CHAPTER 6

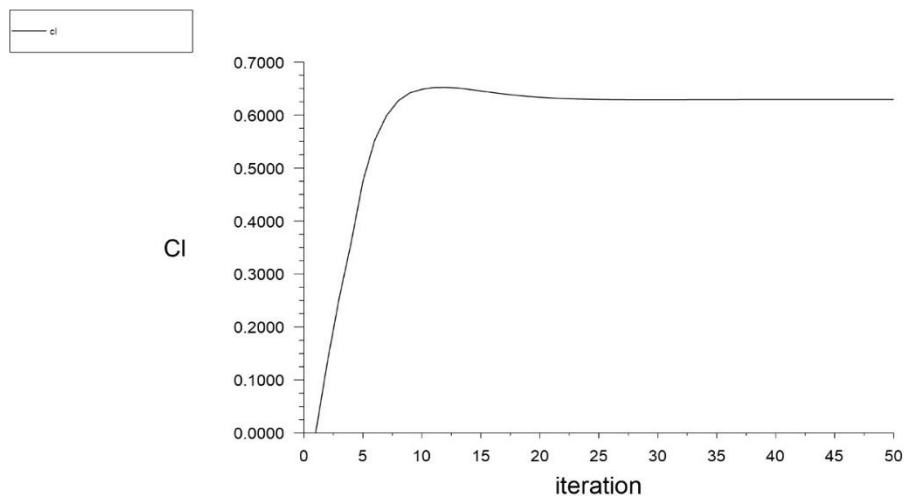
RESULTS AND DISCUSSION

6.1) Preliminary result:

There are two sets of results for the analysis of the problem. The results were obtained for the analysis of the variation of free stream velocity and for the determination of an efficient dimension set up of the shrouded wind turbine using 2-D CFD models of shrouded wind turbine with varying the following parameters as follows.

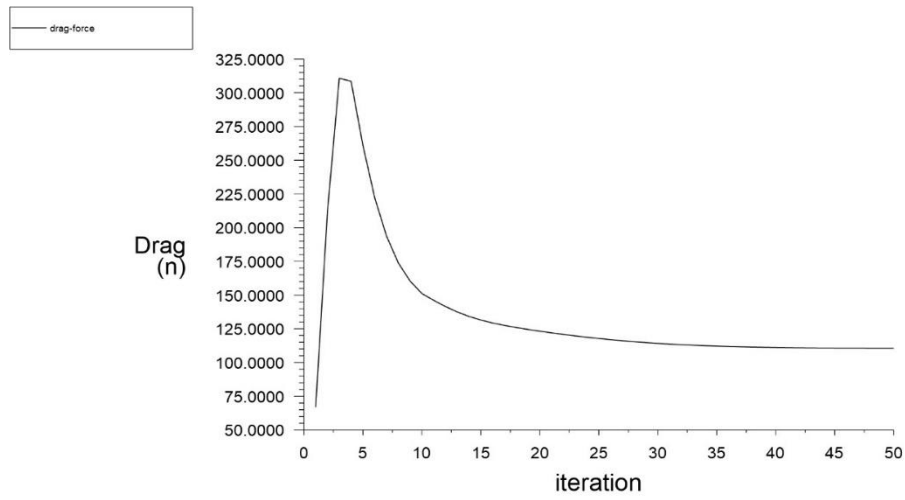
6.1.1) NACA 0018 RESULTS:

By varying the thickness of the brim in the windward direction and maintaining all the other dimensions of the model constant.

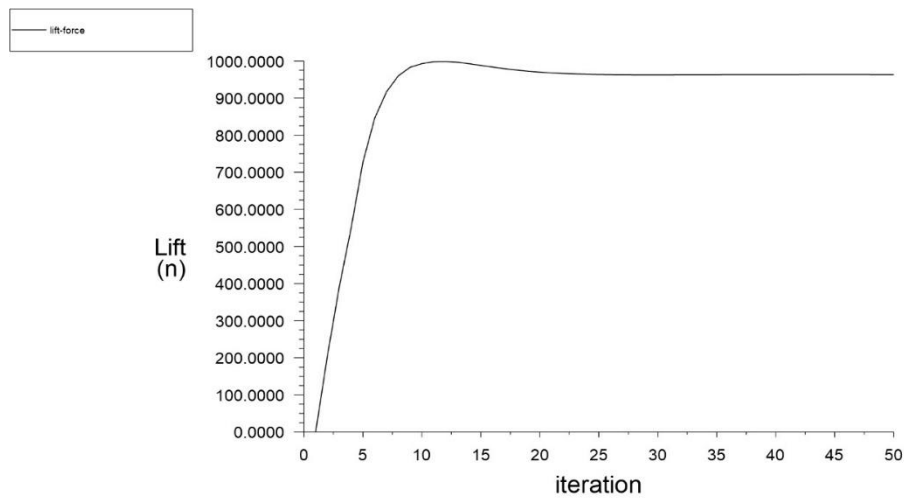


Forces - Direction Vector (1 0 0)				Coefficients		
Zone	Forces (n)			Pressure	Viscous	Total
	Pressure	Viscous	Total			
wall-naca_0018_elevons_-_0_deg	50.801474	59.78445	110.58592	0.033176473	0.039042906	0.072219379

Net	50.801474	59.78445	110.58592	0.033176473	0.039042906	0.072219379



Forces - Direction Vector (0 1 0)						
		Forces (n)		Coefficients		
Zone		Pressure	Viscous	Total	Pressure	Viscous
wall-naca_0018_elevons_-_0_deg	963.62041	-0.20613237	963.41427	0.62930312	-0.00013461706	0.62916851
Net	963.62041	-0.20613237	963.41427	0.62930312	-0.00013461706	0.62916851



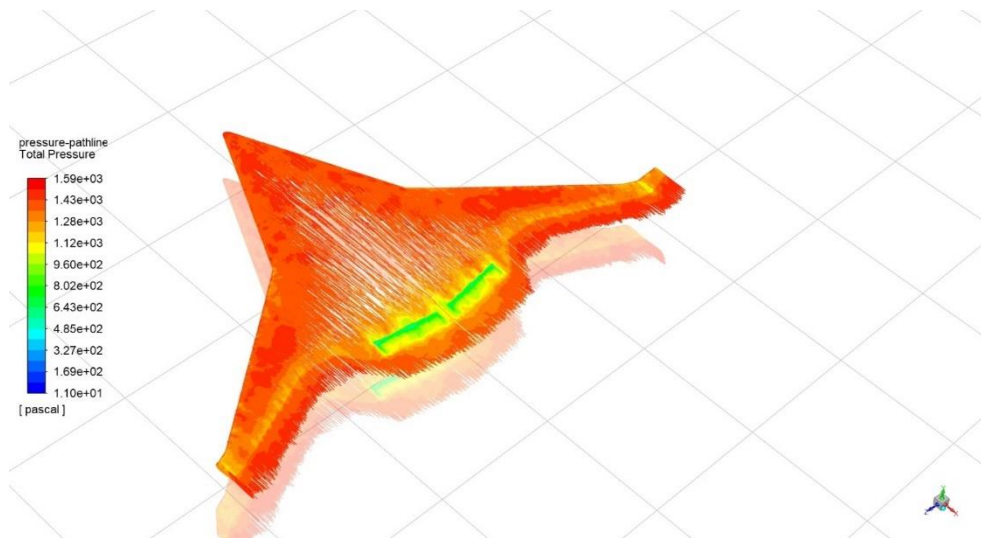


fig.19: Pressure path line of NACA 0018

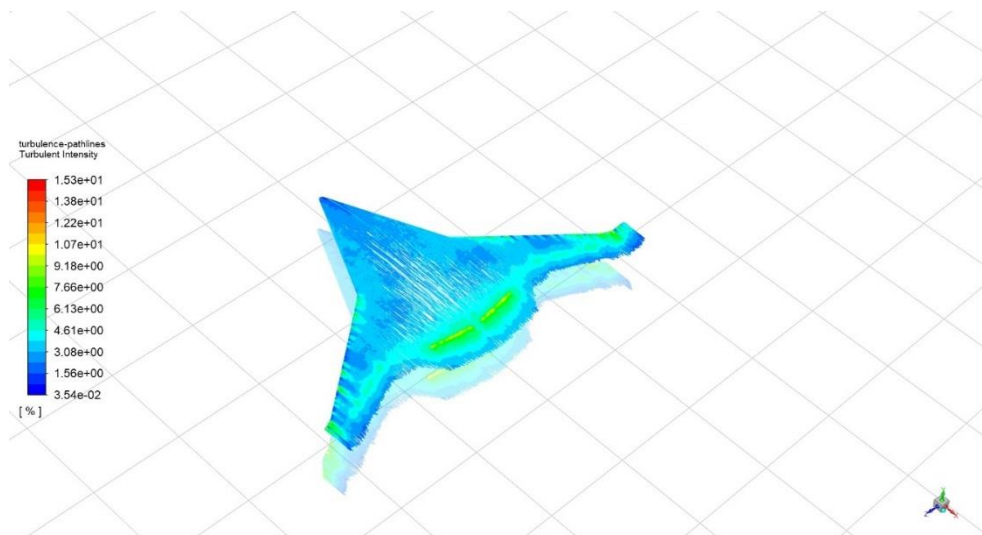
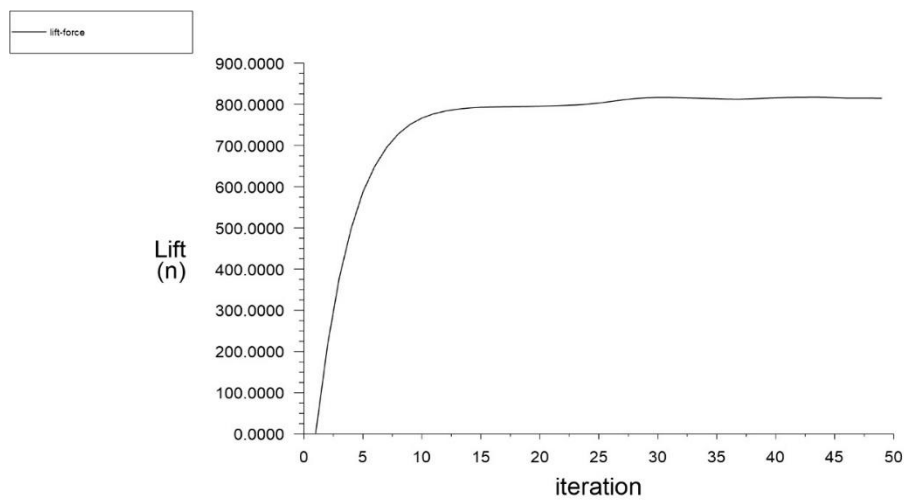
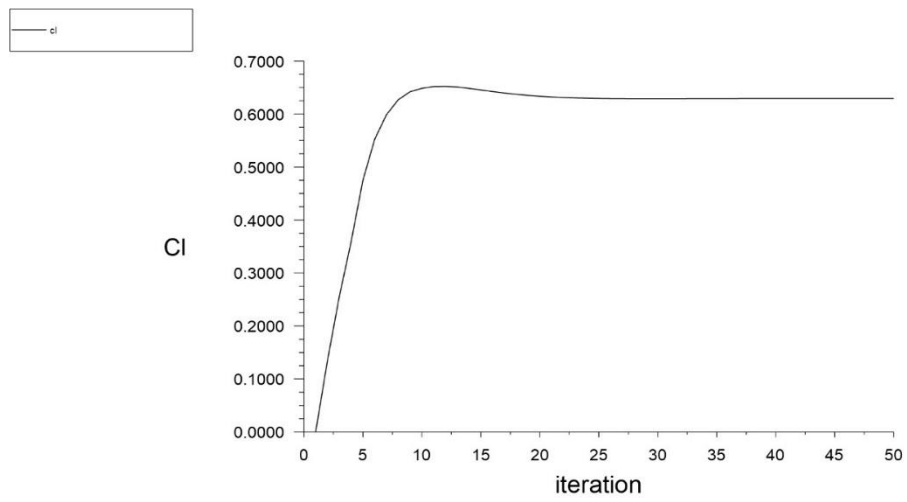


fig.20: Turbulence path line

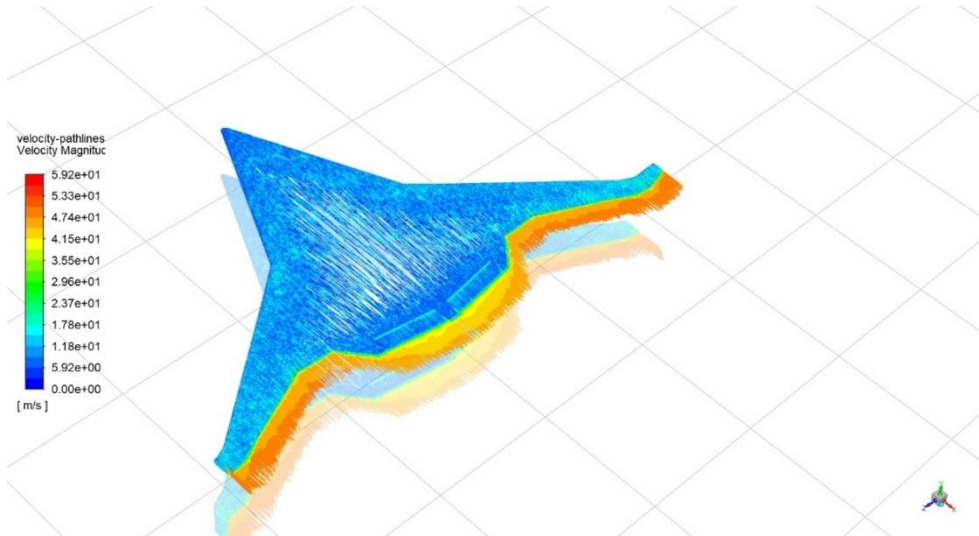


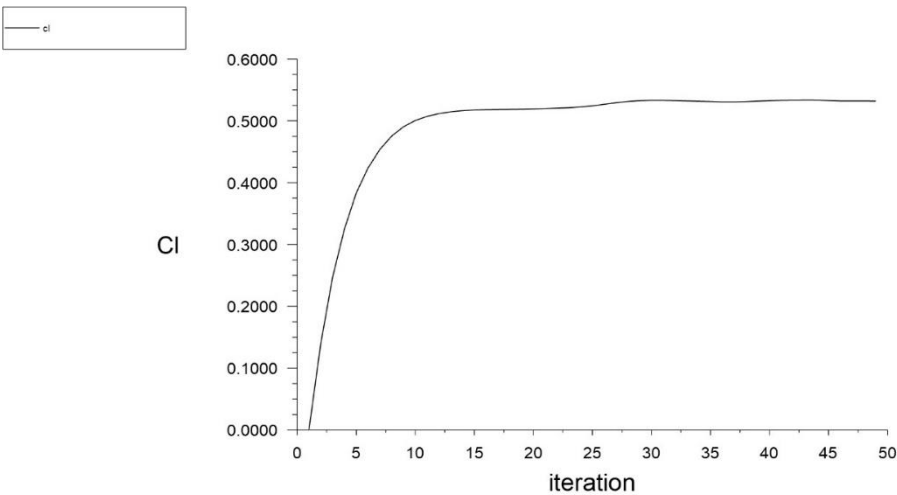
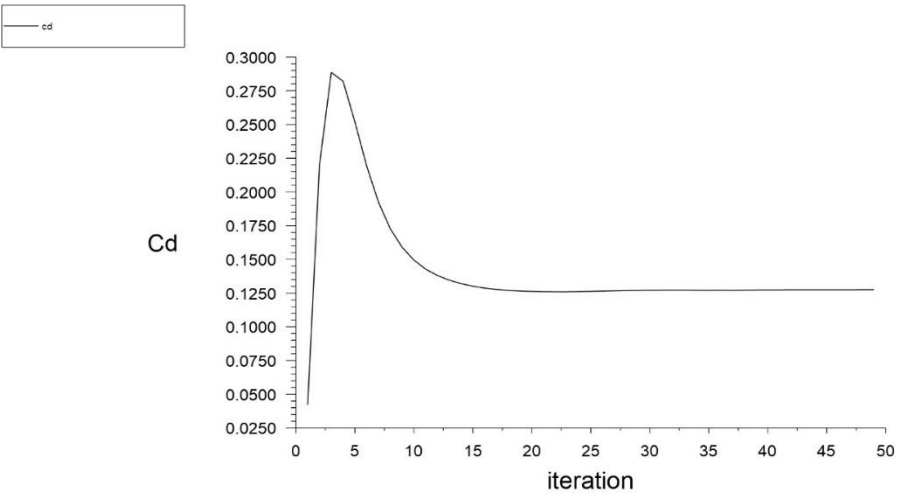
fig.21: Velocity vector for Reference model

6.2) Velocity observation:

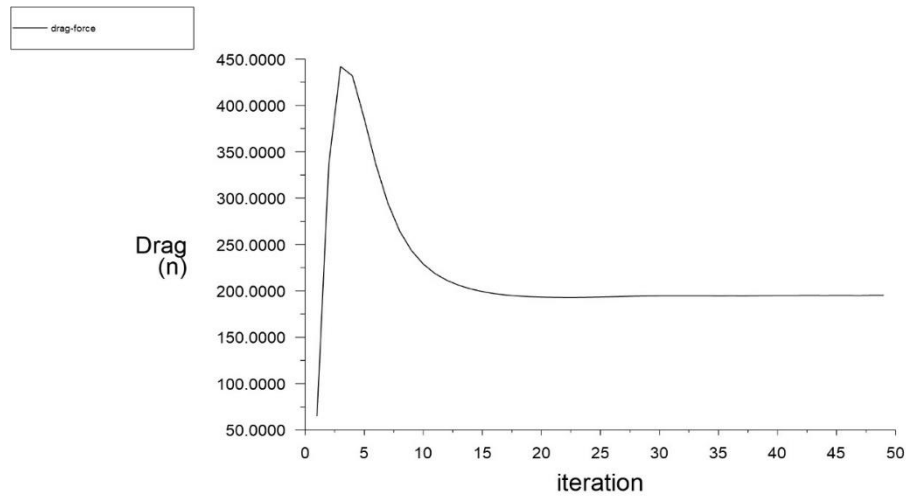
The inlet velocity given is 50m/s. The maximum velocity reached in this case is observed to be 59.2 m/s. The velocity is maximum at the throat section. This is the position where the turbofan would be fixed.

The velocity at the leading edge of the aircraft reaches 0m/s. Figures give a picture of this contour. The Figure gives the visualization of the velocity vector that shows the changes in velocity at different areas in the flow field.

6.3) NACA 0024 Results:



Forces - Direction Vector (1 0 0)						
Zone	Forces (n)			Coefficients		
	Pressure	Viscous	Total	Pressure	Viscous	Total
wall-naca_0018_-_0deg	169.96303	23.653419	193.61645	0.11099627	0.015447131	0.1264434
Net	169.96303	23.653419	193.61645	0.11099627	0.015447131	0.1264434



Forces - Direction Vector (0 1 0)						
Forces (n)						
Zone	Pressure	Viscous	Total	Coefficients Pressure	Viscous	Total
wall-naca_0018_-_0deg	811.68488	-0.18517576	811.49971	0.53007992	-0.00012093111	0.52995899
Net	811.68488	-0.18517576	811.49971	0.53007992	-0.00012093111	0.52995899

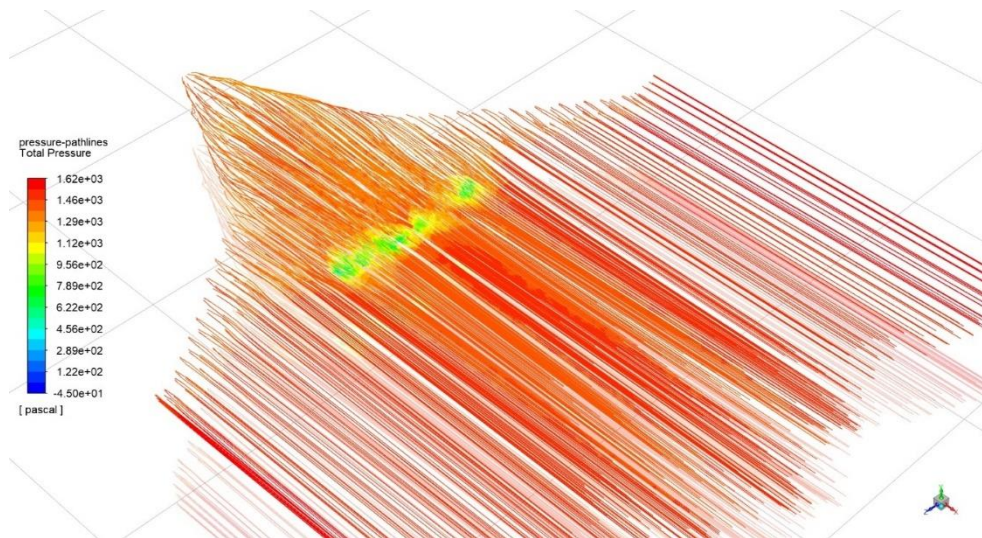
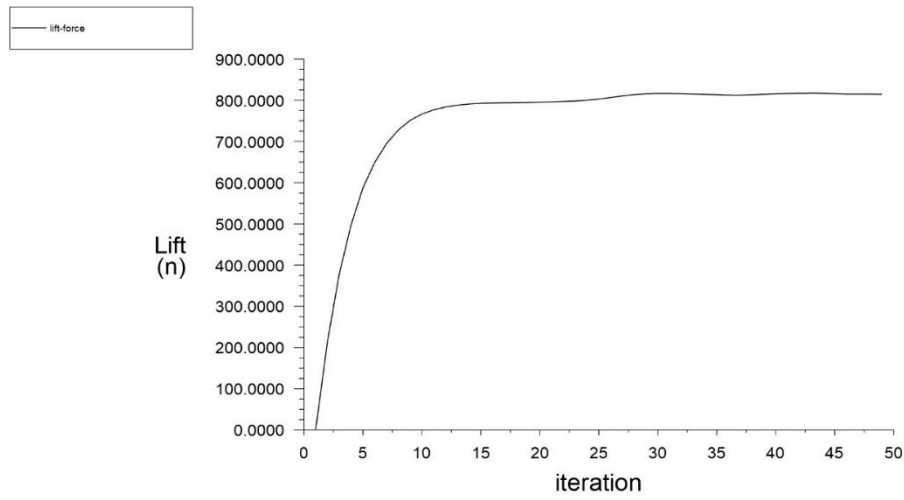


fig.22: Pressure path line for NACA 0024

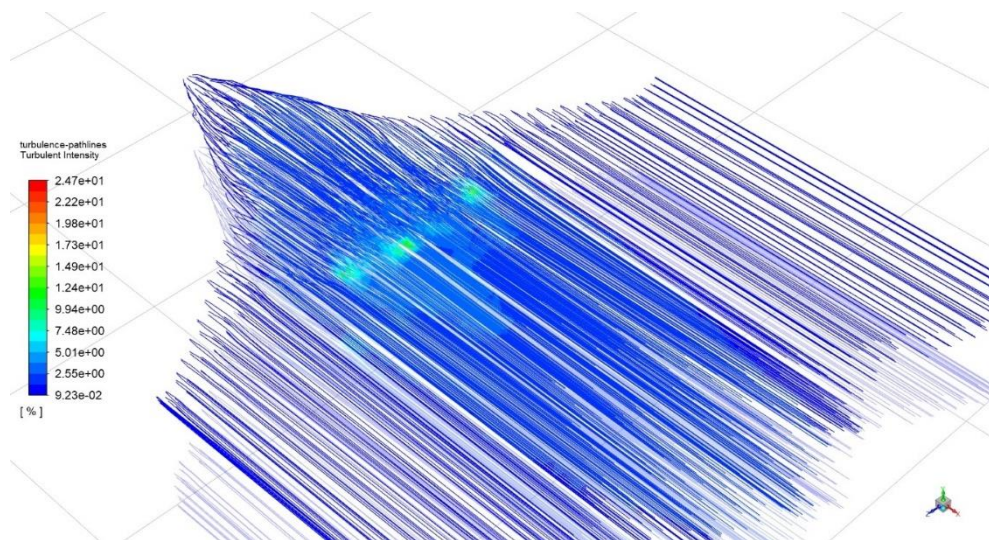
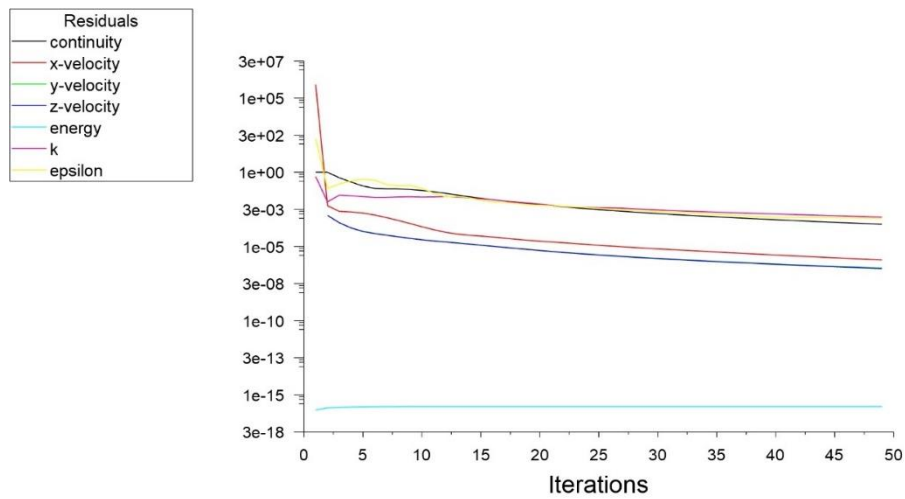


fig.23: Turbulence path line for NACA 0024

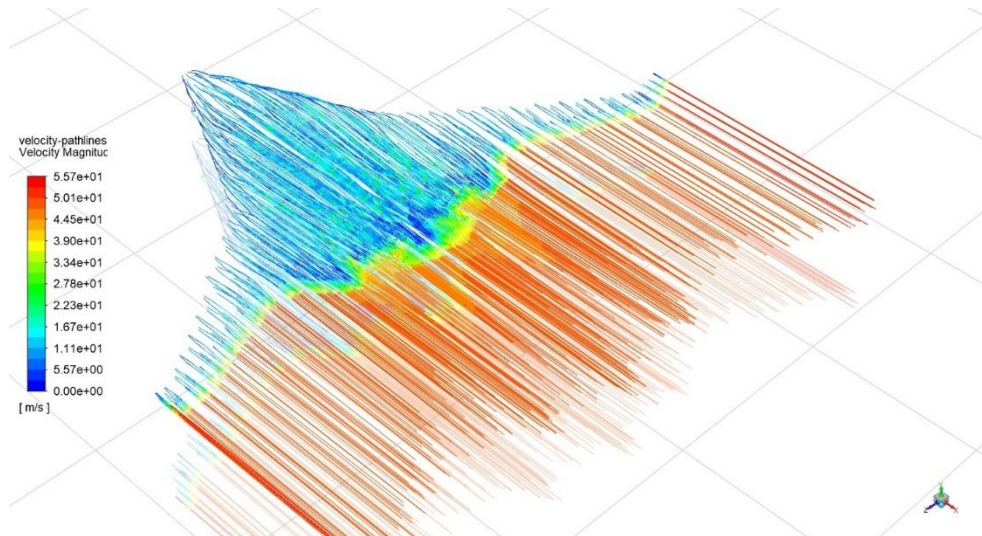


fig.24: Velocity path line for NACA 0024

6.3.1) Velocity observation: The velocity of the free stream is given as 50m/s. It is observed that the maximum velocity of flow is found to be at the throat section. The magnitude of the maximum velocity is 55.7m/s.

The minimum velocity of the flow is at the leading edge 0m/s which is in the region starting from the nose of the aircraft

COMPARISON GRAPH OF AERODYNAMIC EFFICIENCY OF TWO A AIRFOILS (NACA 0018 and NACA 0024)

AIRFOILS	LIFT(L) (N)	DRAG(D) (N)	LIFT COEFFICIENT (L)	DRAG- COEFF(D)	L/D
NACA 0018	963.414	110.585	0.6296	0.0722	8.77119
NACA 0024	811.4997	193.616	0.5299	0.1269	4.1912

6.3.3) Tabulation of aerodynamic efficiency of two airfoils

6.4) RESULTS:

Aircraft with a Higher L/D ratio is more efficient. Among Two NACA airfoils 0018 and NACA 0024. The NACA 0018 has more efficient with a higher L/D ratio of 8.77119. Now we have to perform an aerodynamic analysis of NACA 0018 with different angles of attacks with ELEVEN in Maximum deflection (-35) degree in the counterclockwise direction.

6.4.1) Observations of NACA 0018 with different angles of attacks and Maximum elevon deflection (-35) degree:

ELEVON DEFLECTION (-35) deg and neutral (0) deg VELOCITY PATHLINE

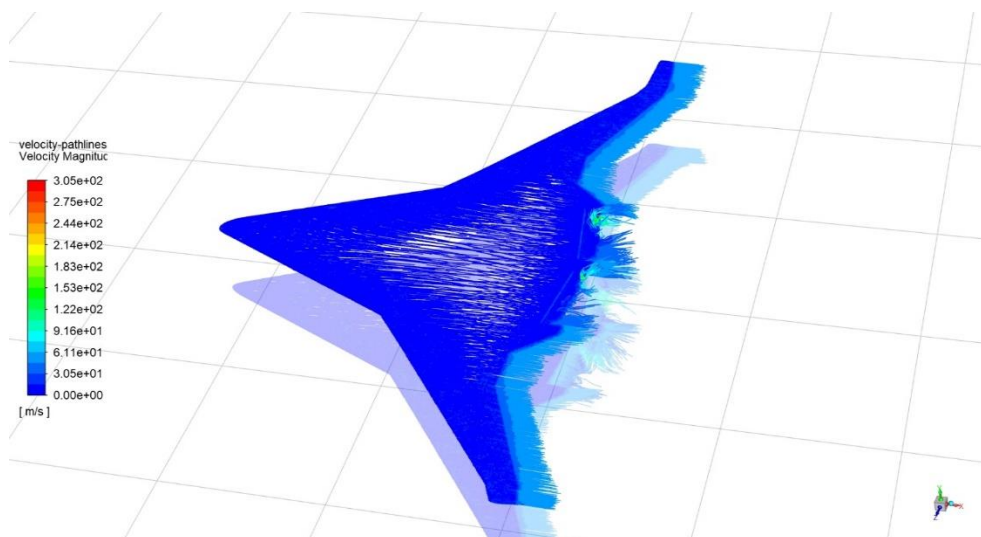


Fig.25: Velocity path line (0, -35) deg

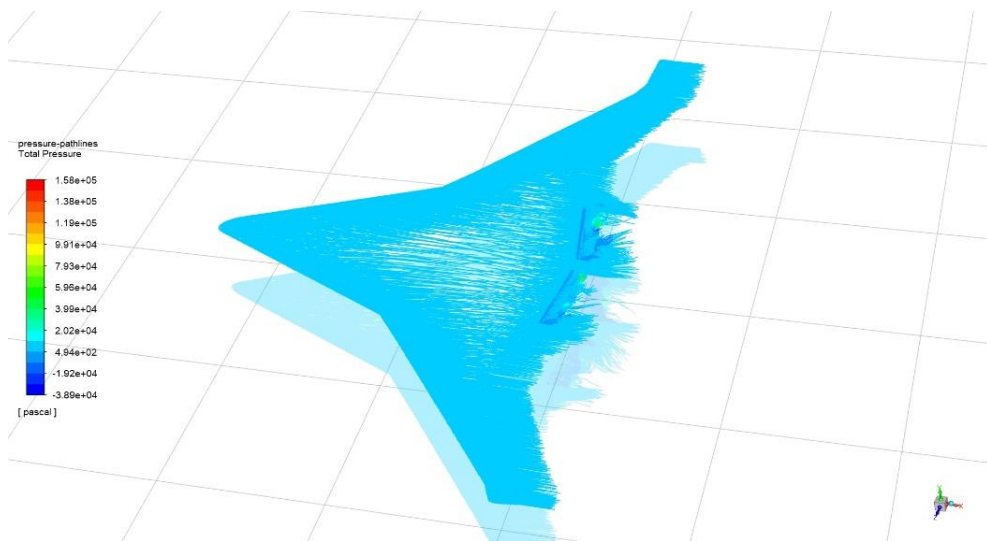
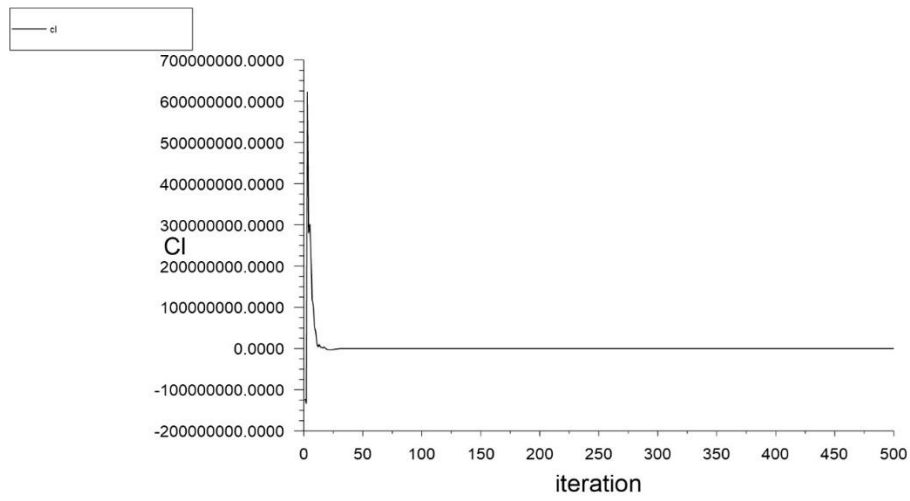
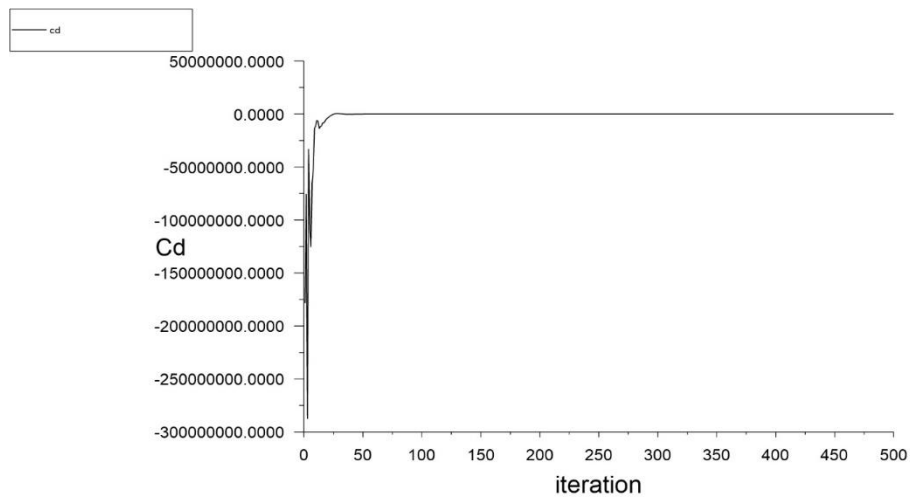


fig.26: Pressure path line (0, -35) deg

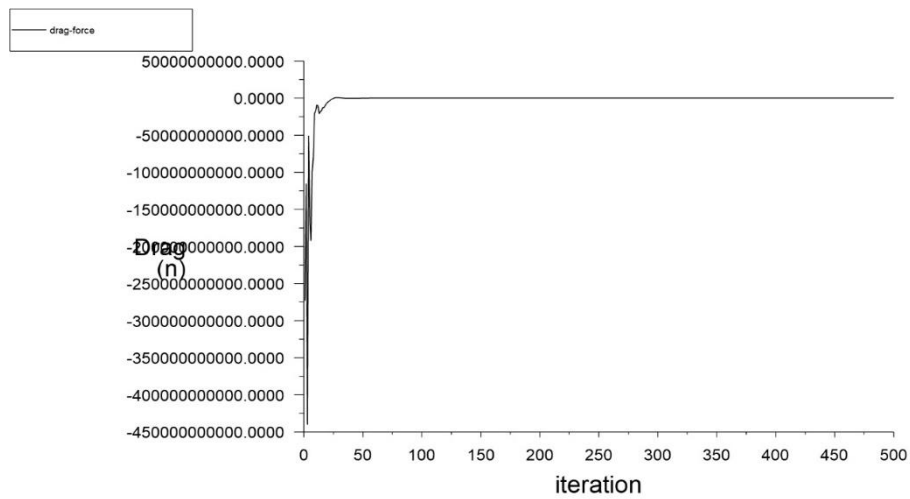
cd		
	Cd	()
wall-takeoff_0_-35		0.46268543



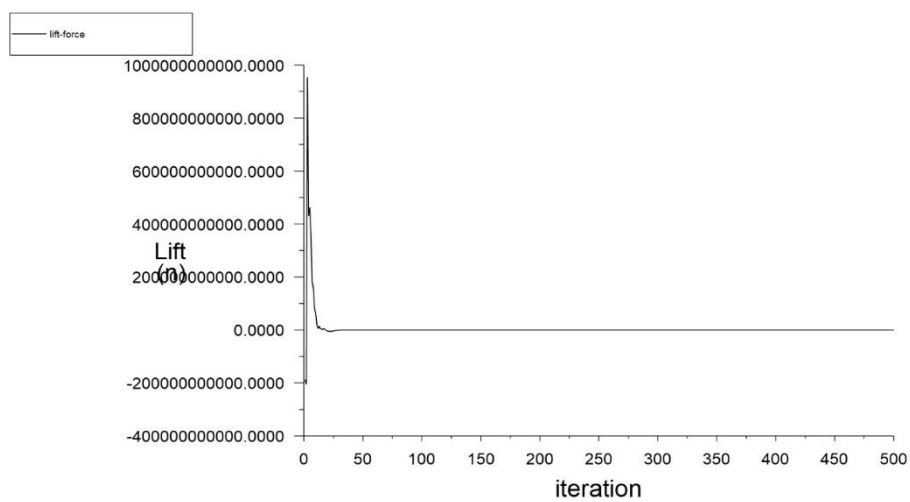
c1	

c1	()

wall-takeoff_0_-35	-1.4565196



drag-force	
Drag	(n)
wall-takeoff_0_-35	708.48707



lift-force	
Lift	(n)
wall-takeoff_0_-35	-2230.2957

6.4.2) ELEVON DEFLECTION -35 DEGREES AND ANGLE OF ATTACK OF 4 DEGREES:

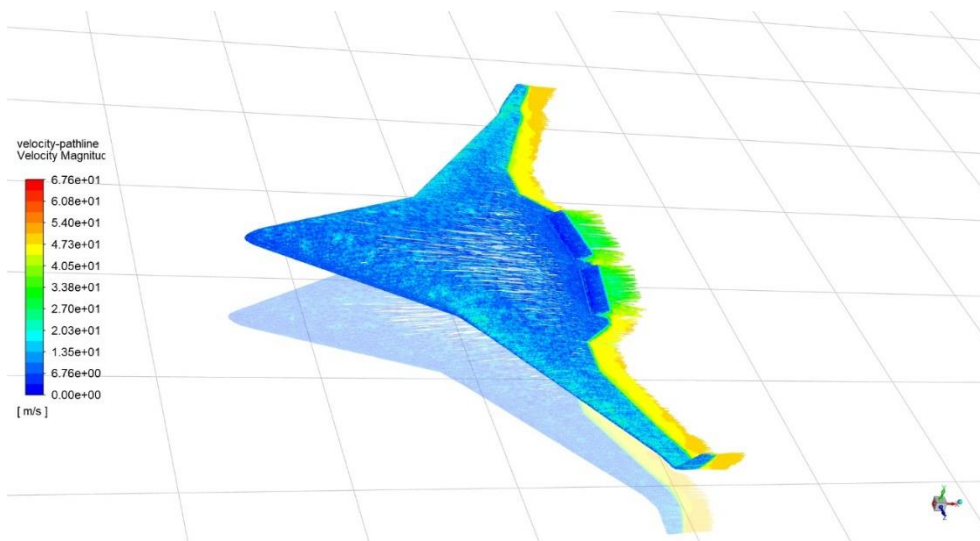


fig.27: Velocity path line (4, -35) deg

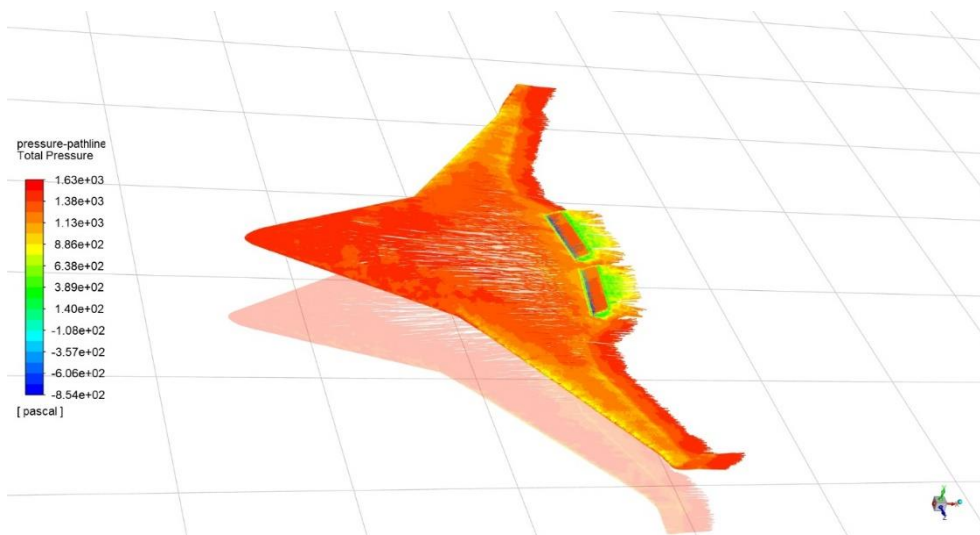
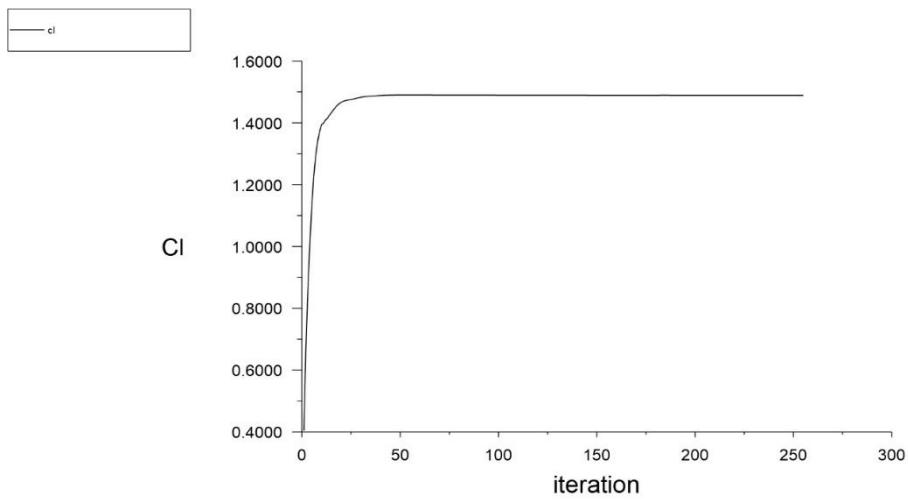
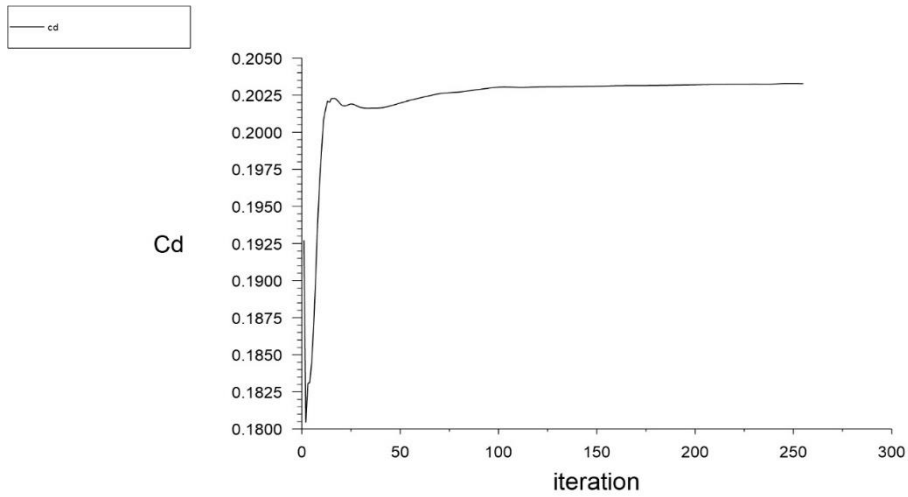


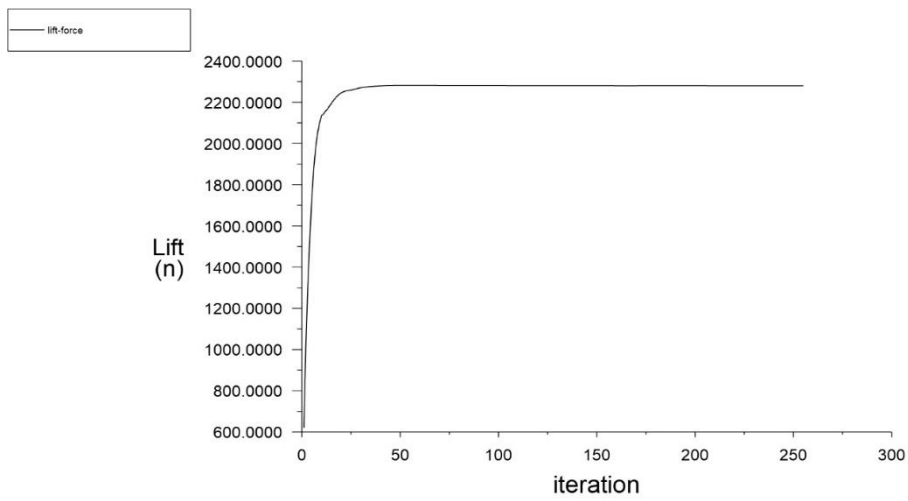
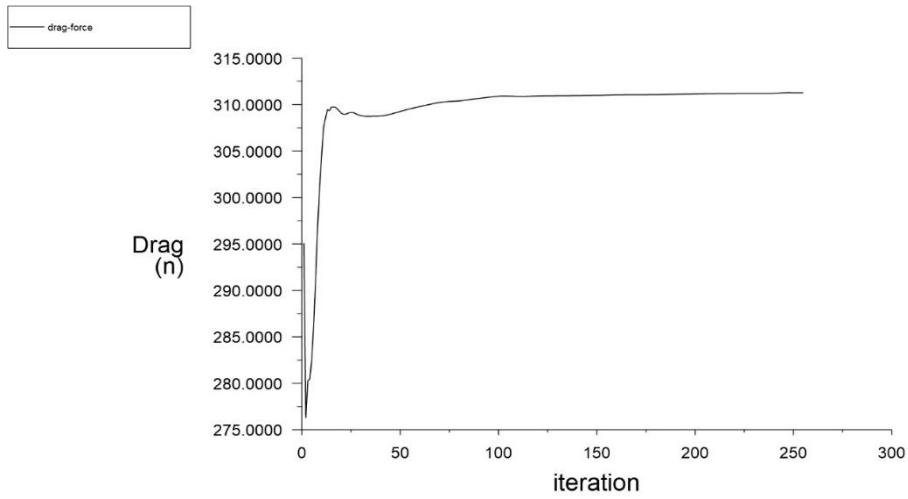
fig.28: Pressure path line (4, -35) deg

cd	
Cd	()
wall-_4_-35_	0.20327332



cl	
Cl	()
wall-_4_-35_	1.4888311

drag-force	
Drag	(n)
wall-4-35_	311.26228



lift-force	
Lift	(n)
wall-4-35_	2279.7726

6.4.3) ELEVON DEFLECTION -35 DEGREES AND ANGLE OF ATTACK OF -4DEGREE:

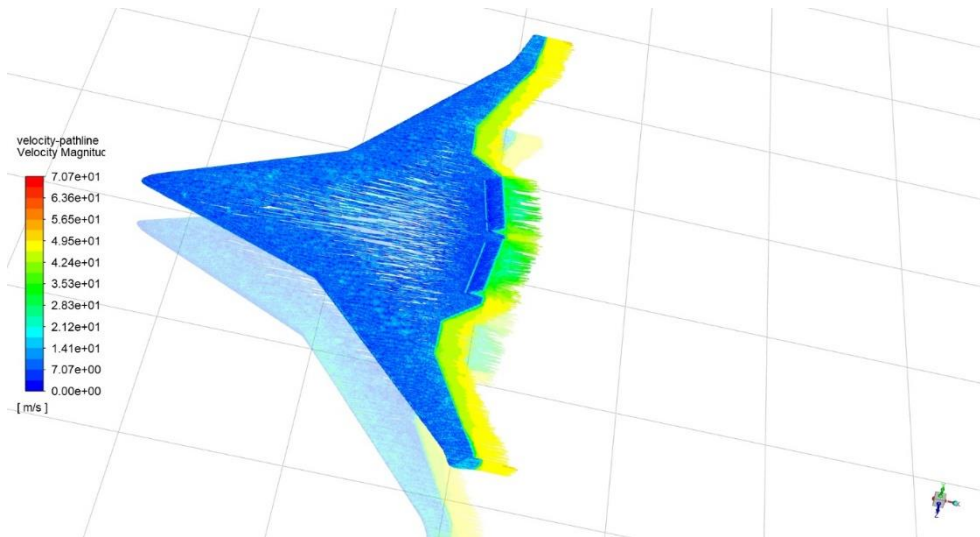


fig.29: Velocity path line (-4, -35) deg

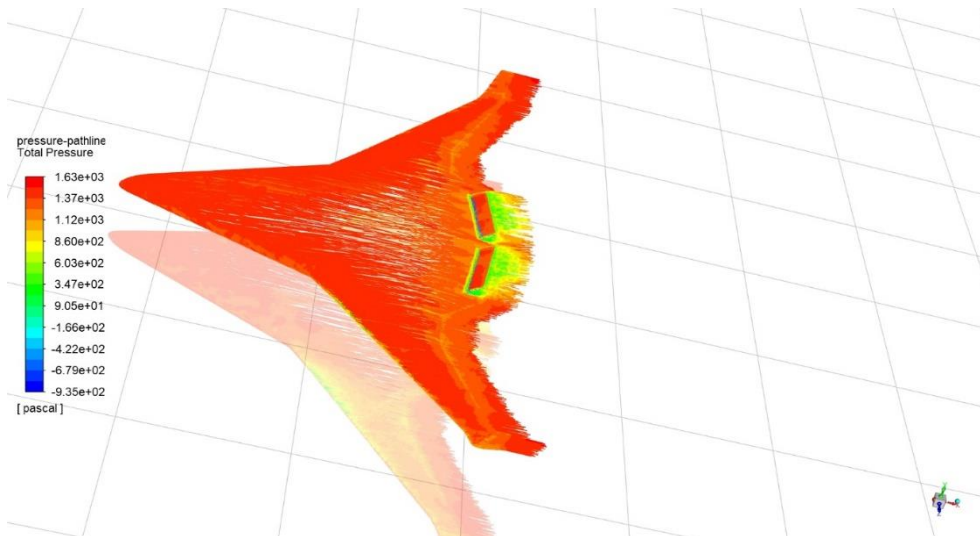
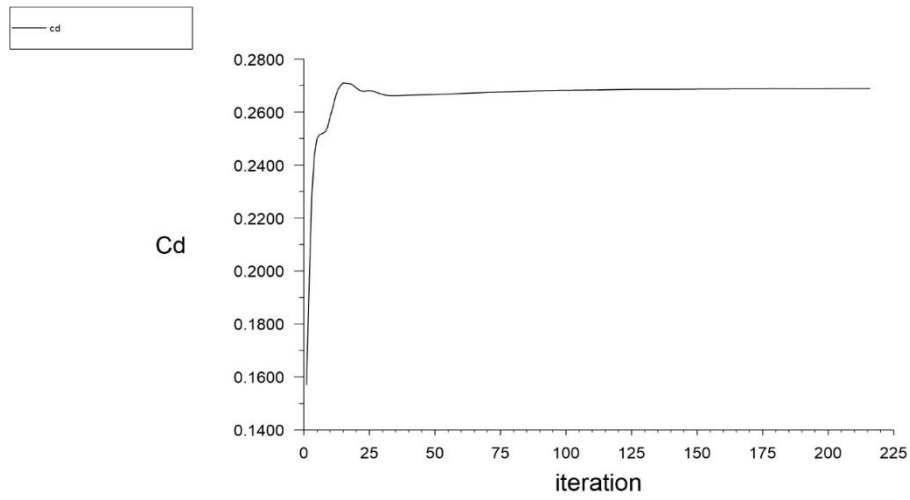
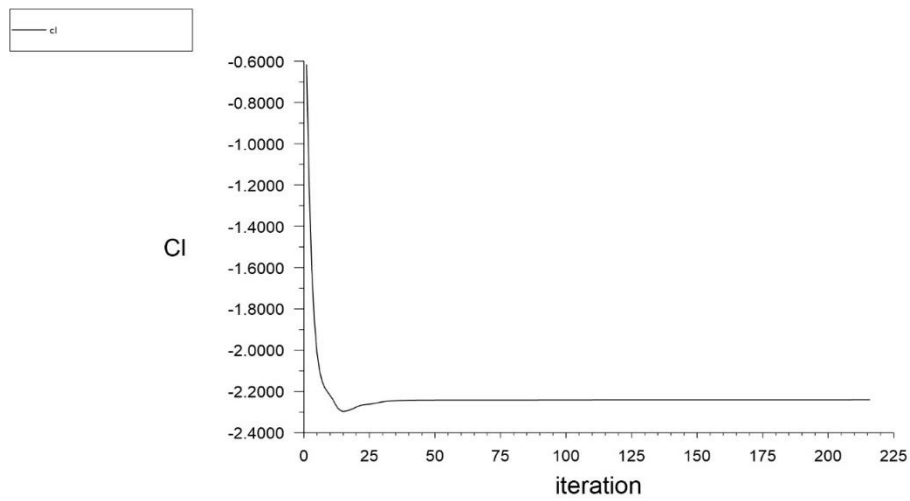


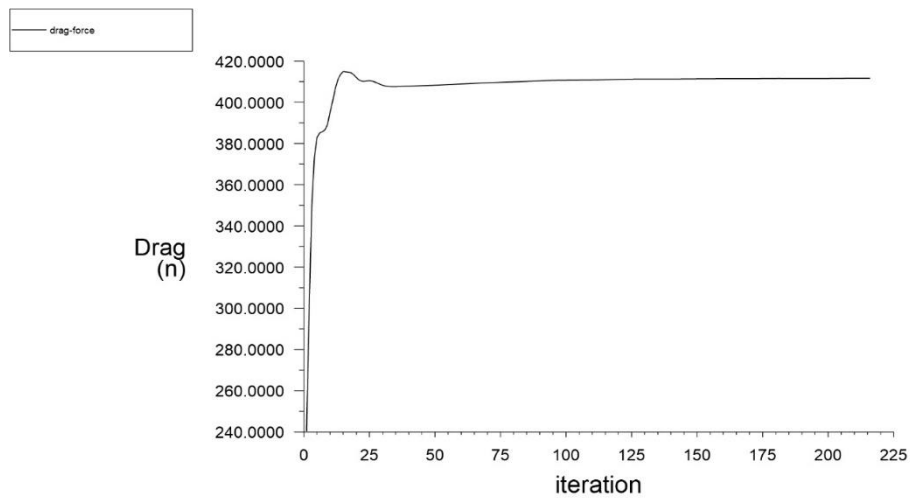
fig.30: Pressure path line (-4, -35) deg



cd		
	Cd	()
wall--4__-35		0.2688054



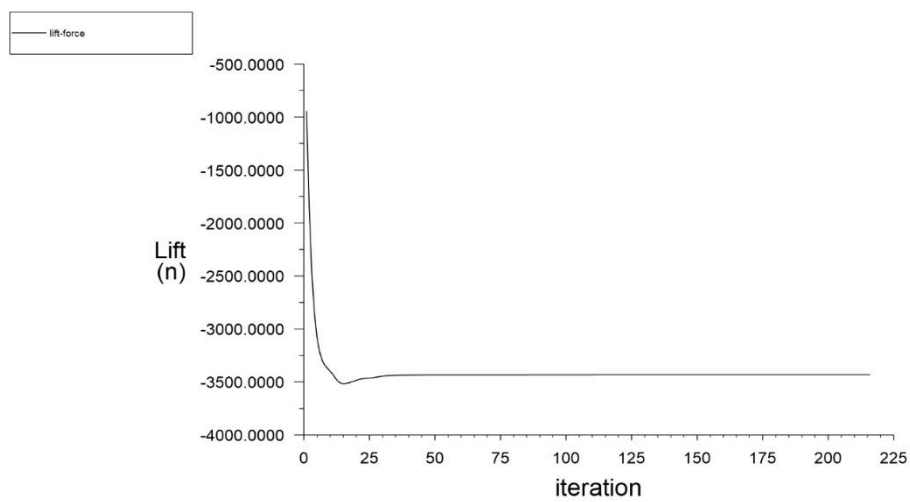
cl		
	Cl	()
wall--4__-35		-2.2401552



drag-force		

	Drag	(n)

wall--4__-35		411.60826



lift-force		

	Lift	(n)

wall--4__-35		-3430.2377

6.4.4) ELEVON DEFLECTION -35 DEGREES AND ANGLE OF ATTACK OF 8 DEGREES:

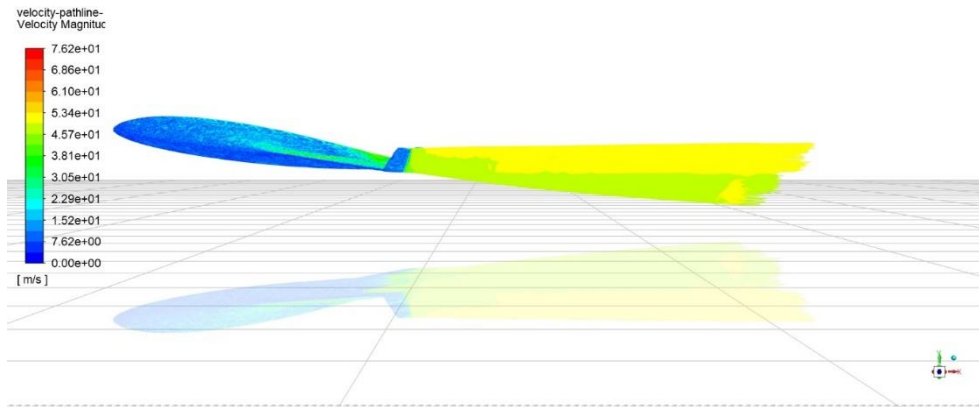
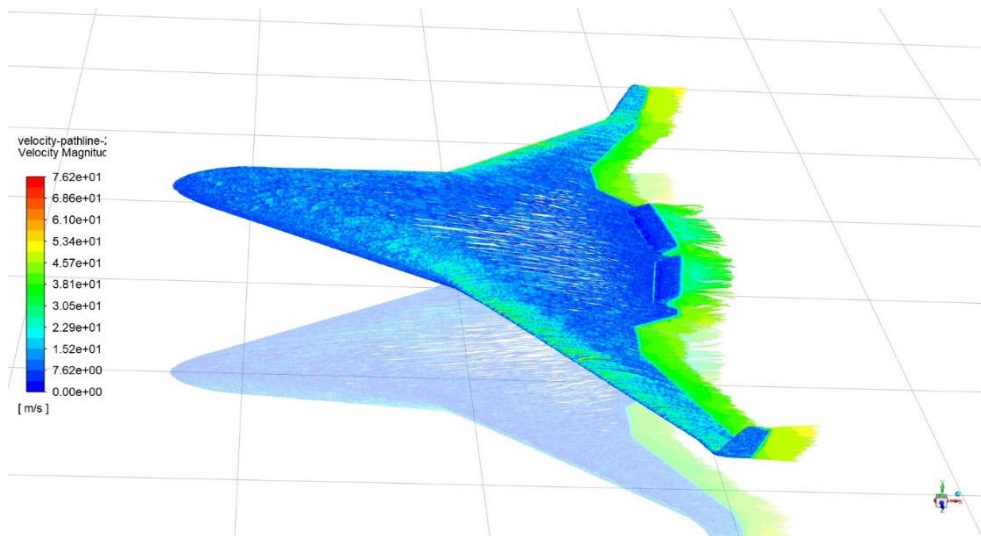


Fig.31: Velocity path line (8, -35) deg



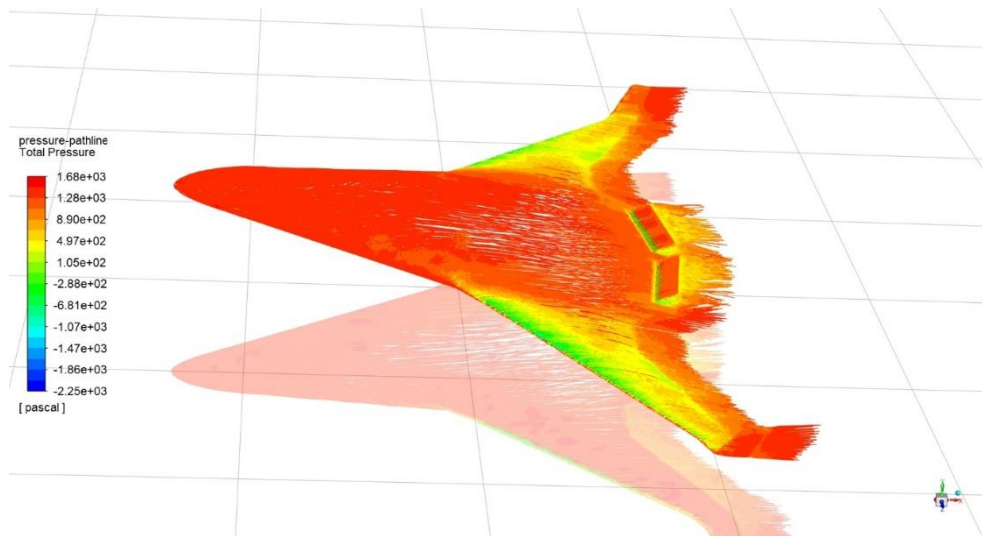
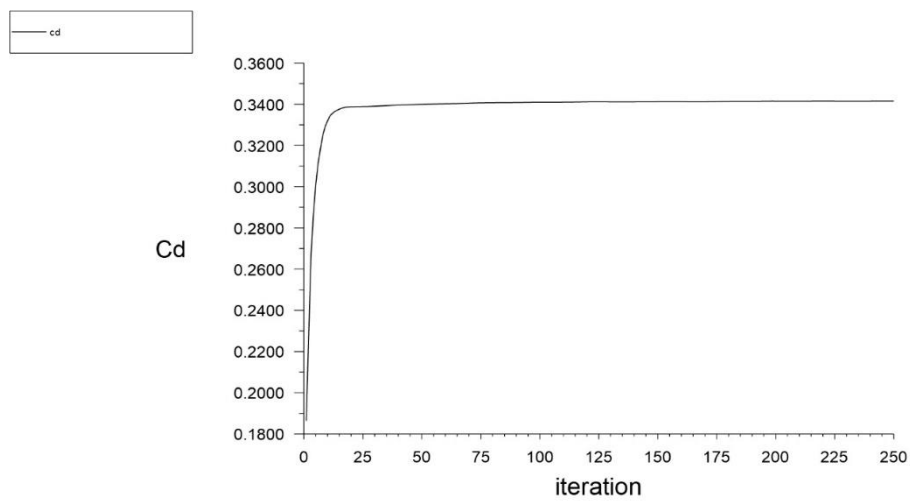
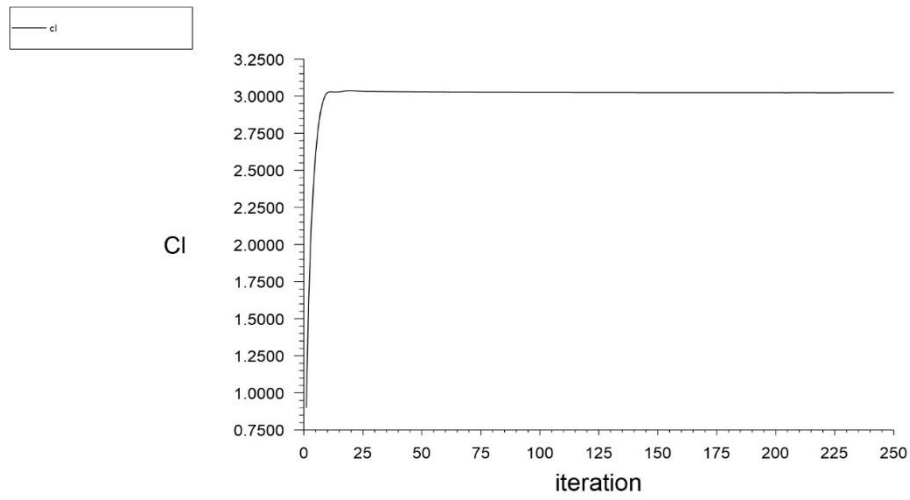


fig.32: Pressure path line (8, -35) deg



cd	
Cd	()
wall_8_-35_	0.3415365



cl	

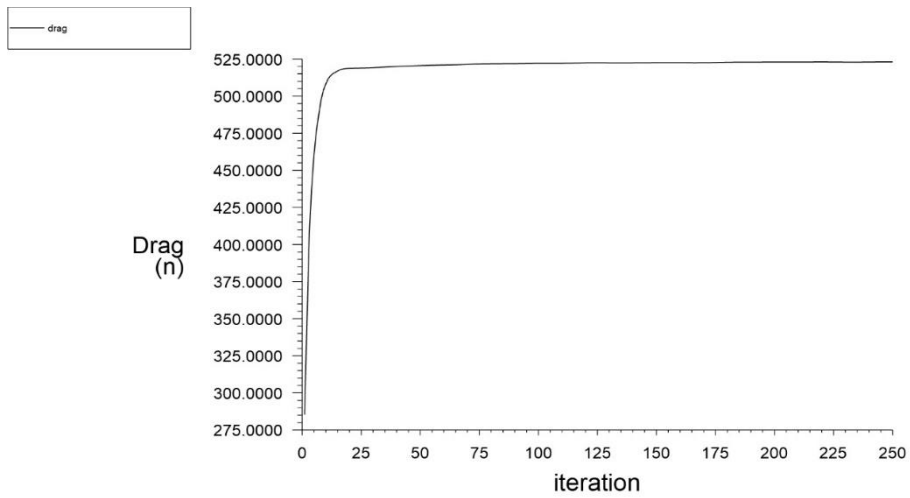
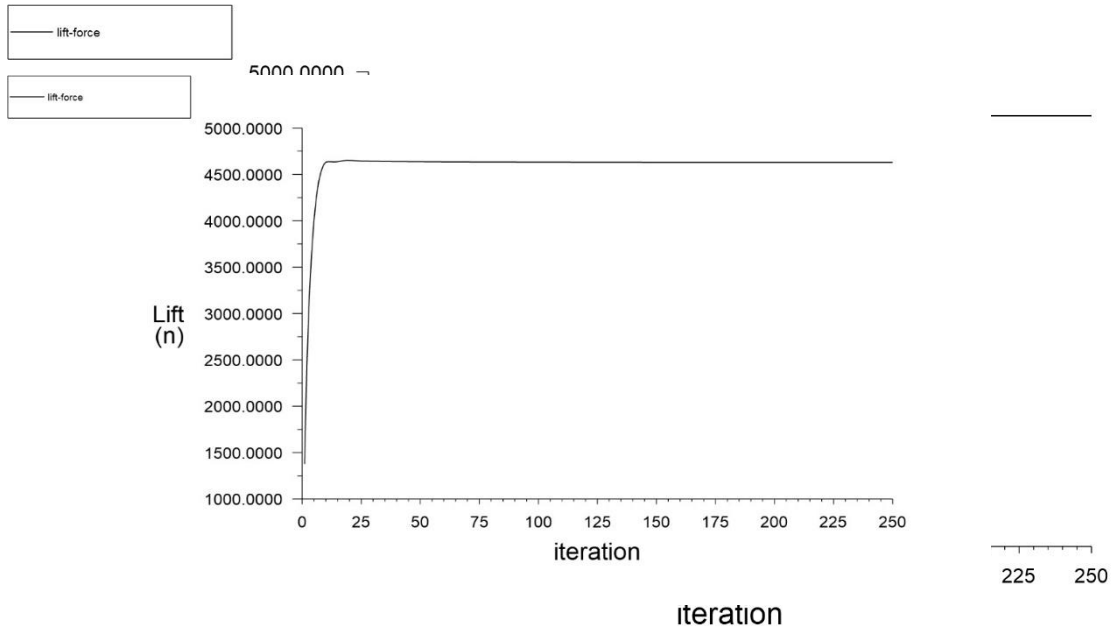
CI	()

wall-8-35_	3.0232383

lift-force	

Lift	(n)

wall_8_-35_	4629.3336



AIRFOIL	FLIGHT SEGMENT	ANGLE OF ATTACK (deg)	ELEVATION DEFLECTIONS (deg)	FREESTREAM VELOCITIES (m/s)	LIFT (N)	DRA G (N)	LIFT COEFFICIENT (CL)	DRA G COEFFICIENT (CD)	LIFT TO DRA G RATIO (L/D)
NACA0018	CRUISE	0	-35	50	2230.295	708.487	1.4565	0.4626	3.147
NACA0018	CRUISE	4	-35	50	2279.772	311.262	1.4888	0.2032	7.324
NACA0018	CRUISE	-4	-35	50	3430.237	411.608	2.2401	0.2688	8.333
NACA0018	CRUISE	8	-35	50	4629.333	522.977	3.0238	0.3415	8.851

6.3.4) Comparison of the values for the NACA0018 with different angles of attack:

6.5) RESULTS: From the calculation table, NACA0018 with different angles of attack namely -4, 0, 4, 8 degrees have shown varied results of aerodynamic parameters values. Among the results, NACA0018 with an angle of attack of 8 degrees has greater aerodynamic efficiency, and less drag produced in the high lift. The aim of this project achieved and produced less drag with a high lift and the lift to drag ratio was higher. Normally the lift to drag ratio is greater resulting in the greater performance of aircraft.

CHAPTER 7

3D DESIGN AND CALCULATION

7.1) 3D DESIGN: The 3D model is created by converting the CATIA design file into STL format and checking the correction. The 3D model is printed by a 3D printer and the material was PLA (polylactic acid). Further, the 3D model is examined and tested under the Open Wind Tunnel. For experimentation purposes, the model is created with eight ports which are used to check the pressure points of the model. where it is connected to the pitot tube to check the value of pressure produced by the model with a velocity is 50 m/ s. The values are noted and calculations should be done and taking the best results and discussing the main result.

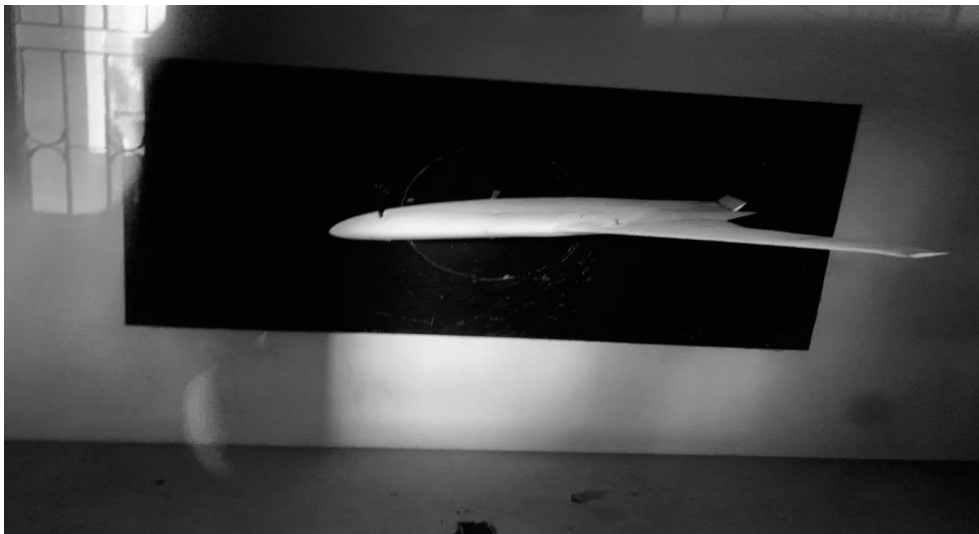


fig.33: Experimentation of 3d model in wind turbine



fig.34: Experimentation of 3d model in a wind turbine with varying angles of attack

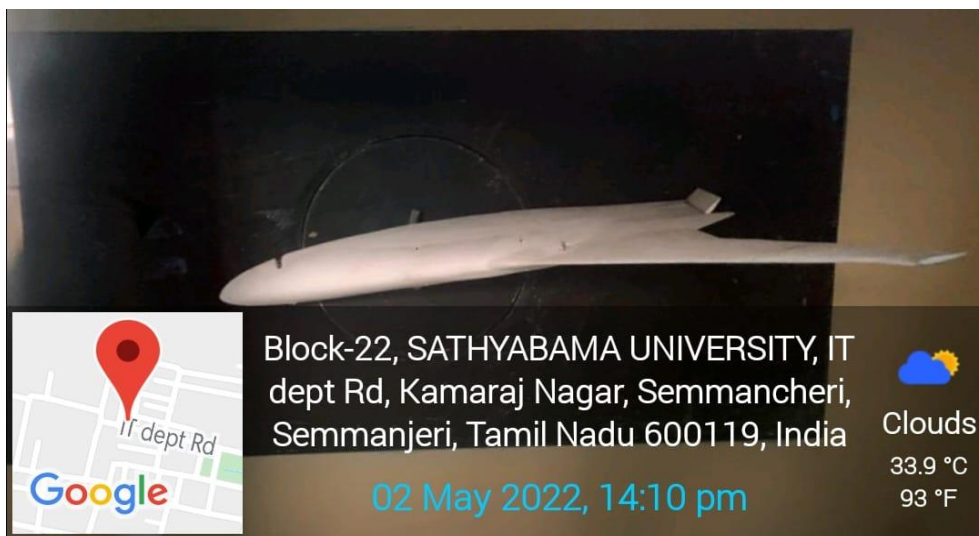
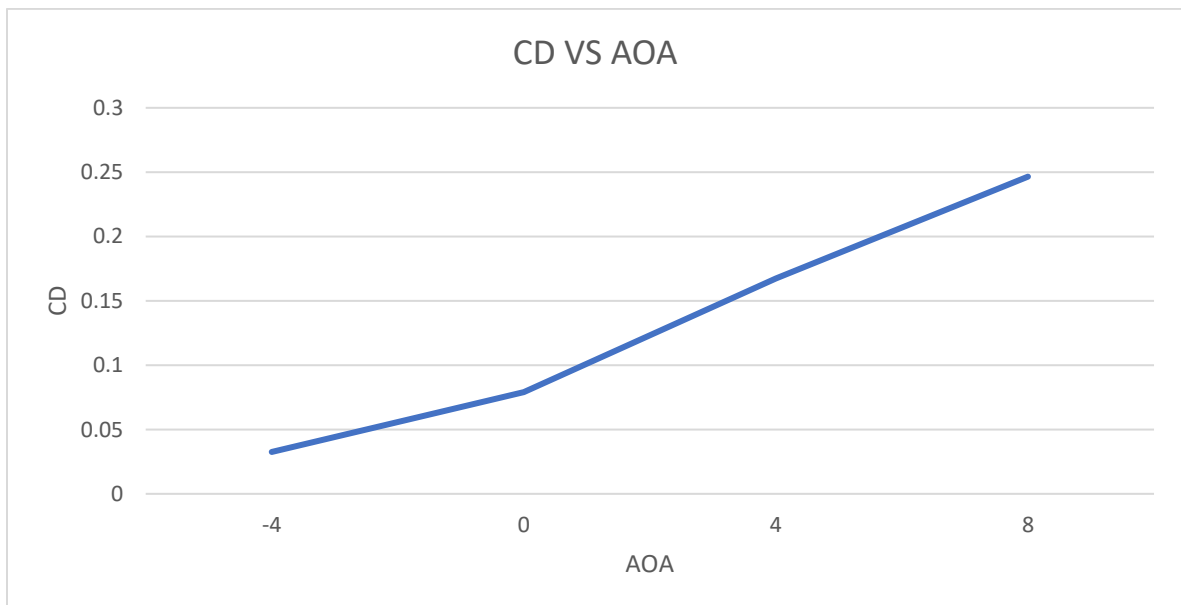
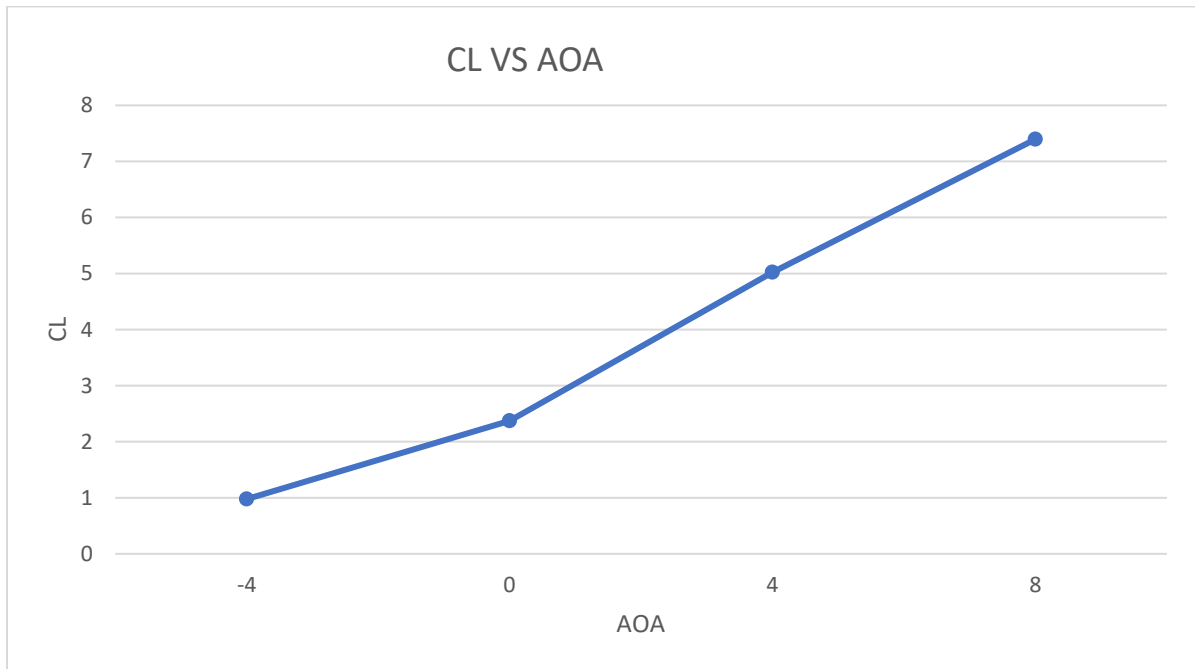


fig.35: Geotag picture of BWB

7.2) CALCULATION:

ANGLE OF ATTACK				COEFFICIENT OF PRESSURE			
-4	0	4	8	-4	0	4	8
3.7	4	4.5	4.8	2.8139	0.4186	0.3023	0.2325
1.5	1.5	1.4	1.5	1	1	1.0232	1
4.1	5.5	7	6.8	0.3953	0.06976	-0.279	-0.2325
4	5.4	6.1	5.9	0.418	0.9301	-0.0697	0.02325
4.8	3.8	3.2	3.1	0.2325	0.4651	0.372	0.6279
3.9	3.1	2.6	2.5	0.4418	0.6271	0.7441	0.7674
1.4	1.5	1.5	1.5	1.0232	1	1	1
2.9	2.8	2.5	2.5	0.6744	0.6976	0.7674	0.767

AOA	CL	CD	CL/CD
-4	0.9768	0.03256	30
0	2.3724	0.079068	30.00455
4	5.0226	0.167432	29.99785
8	7.3952	0.2465	30.00081



CHAPTER 8

CONCLUSION

8.1) Summary and conclusions: Many developments and research have been made over a few decades. The experimentation of BWB with passenger travel was successful finite with some fi and to involves in research" to upgrade the design with limited success. BWB was shown of efficient with a limited number of passengers. An American Newspaper company set that BWB will be a future aviation trend with the development of the new design of BWB Although there is advanced technology and studies to avoid more lift during pitching of aircraft in unison direction is Fluidics and Flexible wing. Fluidics is a study of making use of small interacting flows and fluid jets for function, where forces in aircraft occur due to vortices in which larger complex mechanical pads are changed by smaller simpler fluidic & systems, resulting in a change of direction of aircraft. The main drawback of BWB is a larger wingspan which will occupy more place in the airport. So, the new trend was going to develop the Flexible wing when the wing can change to another size or structure to neglect airflow. X-53 Active Aeroelastic wing is NASA work. The Adaptive Compliant wing is used in military and conventional

References:

6. FLUENT 6.3 (2006) user's Guide. ANSYS Fluent Inc, Lebanon, USA., Murakami, s. (1990) Computational Wind Engineering. Journal of wind Engineering and Industrial Aerodynamics, 36, 517-538.

Soule Hartley A.: Influence of Large Amounts of Wing Sweep on Stability and Control Problem of Aircraft. NACA TN No. 1088, 1946.

Kota, Sridhar; Osborn, Russell; Ervin, Gregory; Maric, Dragan; Flick, Peter; Paul, Donald. Mission adaptive compliant wing-design, fabrication and flight test Ann Arbor, MI; Dayton, OH, U.S.A.: FlexSys Inc., Air Force Research Laboratory. Archived from the original on 22 March 2012. Retrieved 26 April 2011.

Bozic, O, Longo, J.: Moon Sample Return Mission 2016. Proceedings of the 58th International Astronautical Congress, Hyderabad, India, paper IAC-07- A3.6.B.055th (2007).

M. A. Potsdam, M. A. Page and R. H. Liebeck, 1997, "Blended Wing Body Analysis and Design," American Institute of Aeronautics and Astronauts, Inc. AIAA-97-2317.