

A
PROJECT REPORT
ON
**DESIGN AND CFD ANALYSIS OF
BLENDED WING BODY UAV**

SUBMITTED In PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE
AWARD OF THE DEGREE

OF

BACHELOR OF TECHNOLOGY
IN
Aeronautical Engineering



Submitted by:

DHARMENDRA YADAV (16/197)

MUKESH GOKLANI (16/203)

Under the Supervision of:

MR. ANSHUL KHANDELWAL

(Assistant Professor)

Department of Mechanical Engineering

RAJASTHAN TECHNICAL UNIVERSITY

UNIVERSITY DEPARTMENT

KOTA (Rajasthan)

ABSTRACT

Blended wing body aircraft is a fixed-wing aircraft that has a distinct wing and the main body (fuselage) smoothly blended together. It has no clear line separating them. In recent years, there has emerged a significant increase in the interest of the design of BWB aircraft because this design shows significantly improved aerodynamic efficiency compared to conventional aircraft.

In the present study, an attempt has been made to design a Blended Wing Body UAV using SolidWorks, and analysis of it is done through the CFD approach using ANSYS Fluent. The flow pattern, pressure distribution, coefficient of lift, coefficient of drag and lift-to-drag ratio is obtained at velocity 50m/s ($M=0.14$) and Reynolds number of 63,000 for different angles of attack. The results obtained were also promising and cruising lift to drag ratio of design was close to 33.45 which is far better than lift to drag ratio at cruise of any known commercial aircrafts.

DECLARATION

We, the students of the final semester of the mechanical engineering department, University Department, Rajasthan Technical University, Kota declare that work entitled “DESIGN AND CFD ANALYSIS OF BLENDED WING BODY UAV” has been successfully completed under the guidance of Mr. Anshul Khandelwal, Assistant Professor, Mechanical Department. This dissertation work is submitted to University Departments, Rajasthan Technical University in partial fulfillment of the requirements for the award of the degree of bachelor of technology in Aeronautical Engineering.

Dharmendra Yadav (16/197)

Mukesh Goklani (16/203)

CERTIFICATE

This is to certify that the project entitled “DESIGN AND CFD ANALYSIS OF BLENDED WING BODY UAV” submitted by Dharmendra Yadav and Mukesh Goklani in partial fulfillment of the requirements for the award of Bachelor of Technology in Aeronautical engineering at the University Department, Rajasthan Technical University, Kota is an authentic work carried out by them under my supervision and guidance. To the best of my knowledge, the matter embodied in the project has not been submitted to any other university for the award of any degree or diploma.

This report has been prepared as per the prescribed format and is approved submission.

Anshul Khandelwal
Assistant Professor
Department of Mechanical Engineering
Rajasthan Technical University

ACKNOWLEDGMENT

We would like to express sincere thanks to our project guide Sh. Anshul Khandelwal for his valuable guidance and advice throughout the project work, for stimulating our interest in this topic, and for his generous support and constant encouragement through the study. Working on this project helped us a lot in learning about Blended Wing Body aircraft. Finally, we would like to thank our parents and our friends who kept our soul uplifted and cheered us up along the way.

Table of Contents

Abstract	ERROR! BOOKMARK NOT DEFINED. V
Declaration	ERROR! BOOKMARK NOT DEFINED. VV
Certificate	ERROR! BOOKMARK NOT DEFINED. VVV
Acknowledgement	ERROR! BOOKMARK NOT DEFINED. IV
List Of Figures	VII
List Of Tables	VII
Chapter 1: Introduction	1
1.1 Blended Wing Body	1
1.2 Formulation Of BWB Concept	2
1.3 Comparison Of Different Loads	5
1.4 Key Concepts Of BWB	7
1.5 Advantages Of BWB	8
Chapter 2: Computational Fluid Dynamics	10
2.1 Introduction	10
2.2 Uses Of CFD	10
2.3 CFD Methodology	12
2.3.1 Creating Geometry Mesh	13
2.3.2 Defining Of Physics Of Model	13
2.3.3 Solving CFD Problems	14
2.4 Discretization Methods	14
Chapter 3: Software Packages Used	16
3.1 SOLIDWORKS	16
3.1.1 SolidWorks In Aerospace	17
3.2 ANSYS	18
3.2.1 ANSYS Fluent	19
Chapter 4: Design Process	20
4.1 AIRFOIL SELECTION	21
4.2 GEOMETRIC PARAMETERS	22
4.3 DESIGN PROCESS	24
Chapter 5: Meshing Process	26
5.1 ANSYS DESIGN MODELLER AND MESHING	26
5.1.1 Geometric Modelling	26

5.1.2 Creation Of Fluid Domain	27
5.2 MESHING	28
5.2.1 Meshing Fluid Domain	28
5.2.2 Insertion Of Named Section	29
Chapter 6: Results And Discussions	30
Chapter 7: Conclusion	32
7.1 CONCLUSION	32
References:	34

LIST OF FIGURES

Figure 1.1 A Blended Wing Body Aircraft.....	1
Figure 1.2 Aircraft Design Evolution.....	2
Figure 1.3 A Preliminary Concept.....	3
Figure 1.4 Early Blended Wing Concept	3
Figure 1.5 Three Conical Forms.....	4
Figure 1.6 Cylindrical Fuselage Wing Geometry.....	5
Figure 1.7 Total Difference In The Wetted Area Of 10,200 Ft	5
Figure 1.8 Total Wetted Area Difference Of 14,300 Ft.....	6
Figure 1.9 Structural Loading Of The BWB With Conventional Configuration.....	7
Figure 1.10 Blended Wing Body Uav	8
Figure 3.1 Assembly Drawing In SolidWorks	18
Figure 3.2 Ansys Fluent.....	19
Figure 4.1 Mh -45 Airfoil	21
Figure 4.2 Importing Of Airfoil In SolidWorks	24
Figure 4.3 Resizing And Projection Of Airfoil	24
Figure 4.4 Creating Projection Of Wing Tip.....	25
Figure 4.5 Creating Loft(Blend)	26
Figure 4.6 Complete BWB Design.....	26
Figure 5.1 Fluid Domain	27
Figure 5.2 Surrounding Meshing	27
Figure 5.3 BWB Model Meshing.....	28
Figure 5.4 Inlet, Outlet, Wall	30
Figure 6.4 Pressure Distribution Over BWB(0° Aoa)	32
Figure 6.5 Static Pressure Over Center Plane(0° Aoa).....	32
Figure 6.6 Velocity Distribution Over Center Plane(0° Aoa).....	33
Figure 6.7 Variation Of C_l With Aoa.....	34
Figure 6.8 Variation Of C_d With C_d	34
Figure 6.9 Variation Of L/D With Aoa	35

LIST OF TABLES

Table 1 Airfoil Characteristics.....20

Table 2 Geometric Details of Design.....23

Table 3 Force and its Coefficients.....29

Table 4 Lift to Drag Ratios of Various Aircraft.....33

CHAPTER 1

INTRODUCTION

1.1 Blended Wing Body

Blended wing body or Hybrid Wing Body aircraft have a flattened and airfoil-shaped body, where the fuselage is merged with wing and tail to form a single entity. BWB is a hybrid of flying-wing aircraft and the conventional aircraft where the body is designed to have the shape of an airfoil and is carefully streamlined with the wing to have the desired problem.

If the wing in conventional aircraft is the main contributor to the generation of lift, the fuselage of BWB lifts together with the wing, thus increasing the effective lifting surface area. The streamlined shape between the fuselage and the wing intersections reduces interference drag, reduces the wetted surface area that reduces friction drag, while the show evolution of the fuselage to wing thickness by careful design may suggest that more volume can be stored inside the BWB aircraft, hence increased payload and fuel capacity.



Figure 1.1: A Blended Wing Body Aircraft

The BWB concepts aim at combining the advantages of a flying wing with the loading capabilities of a conventional airliner by creating a wide body in the center of the wing to allow space for passenger and cargo. Especially, for very large transport aircraft, the BWB concept is often claimed to be superior compared to

the conventional configuration in terms of higher lift to drag ratio and consequently less fuel consumption.

1.2 Formulation of the BWB Concept

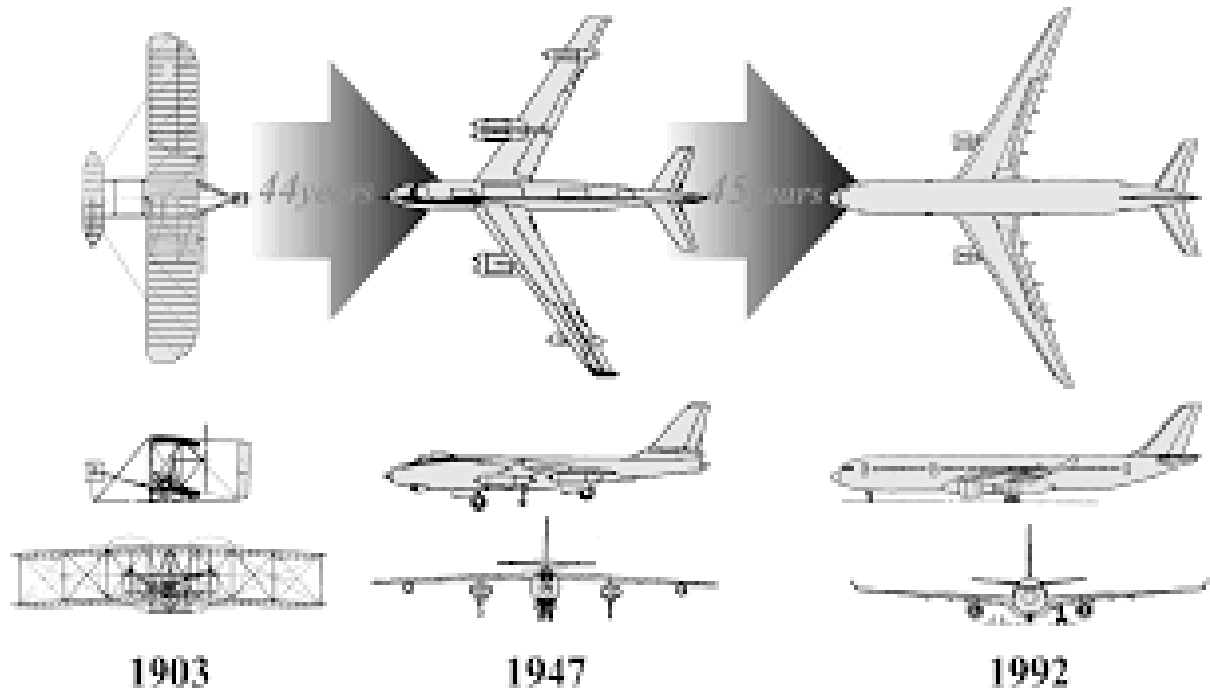


Figure 1.2: Aircraft Design Evolution

It is appropriate, to begin with, a reference to the Wright Flyer itself. Designed and first flown in 1903. A short 44 years later, the swept-wing Boeing 4-47 took flight. A comparison of these two airplanes shows a remarkable engineering accomplishment within a period of slightly more than four decades. Embodied in the B-47 are most of the fundamental design features of a modern subsonic jet transport swept wing and empennage and podded engines hung on the pylons beneath and forward of the wing. The Airbus A330, designed 44 years after the B-47 appears to be essentially equivalent, as shown in figure 1.2.

A preliminary concept is shown in figure 1.3 was the result. Here the pressurized passenger compartment consisted of adjacent parallel tubes, a lateral extension of the double- bubble concept. Comparison with the conventional configuration airplane sized for the same design mission indicated that the blended configuration was significantly lighter, had a higher lift-to-drag ratio, and had a substantially lower fuel burn.

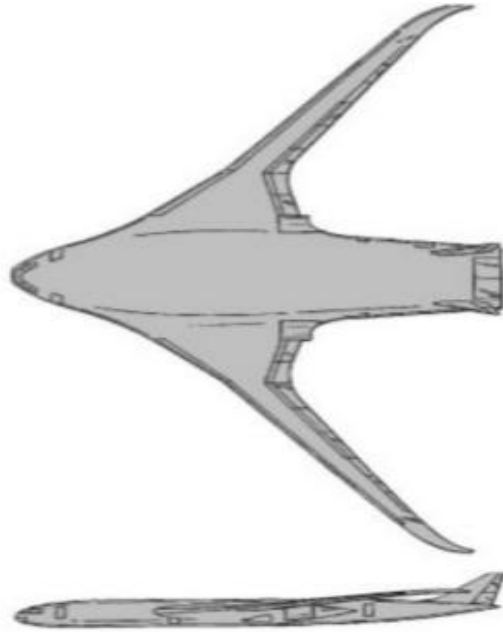


Figure 1.3: A Preliminary Configuration

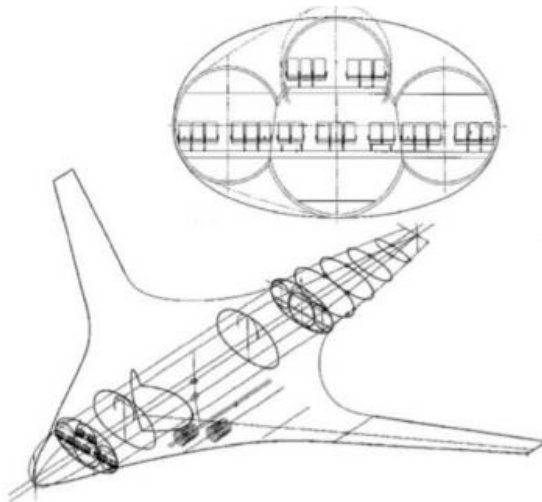


Figure 1.4: Early Blended Wing Concept

The performance potential implied by the blended configuration provided the incentive for NASA Langley Research Center to fund a small study at McDonnell Douglas to develop and compare advance technology subsonic transports for the design missions of 800 passengers and a 7000-n mile range at the mac number of 0.85 composites structure and advance technology turbofans were utilized.

Defining the pressurized passenger cabin for a very large airplane offers two challenges First, the square-cube law shows that the cabin surface area per passengers available for the emergency egress decreases with the increasing passenger count. Second, cabin pressure loads are the most taken in hoop

tension. Thus, the early study began with an attempt to use circular cylinders for the fuselage pressure vessel as shown in fig 1.4 along with the corresponding first cut at the airplane geometry. The engines are buried in the wing root and it was intended that passengers could egress from the sides of both the upper and lower levels. The concept was headed back to a conventional tube and wing configuration. Therefore, it was decided to abandon the requirement for taking BWB.

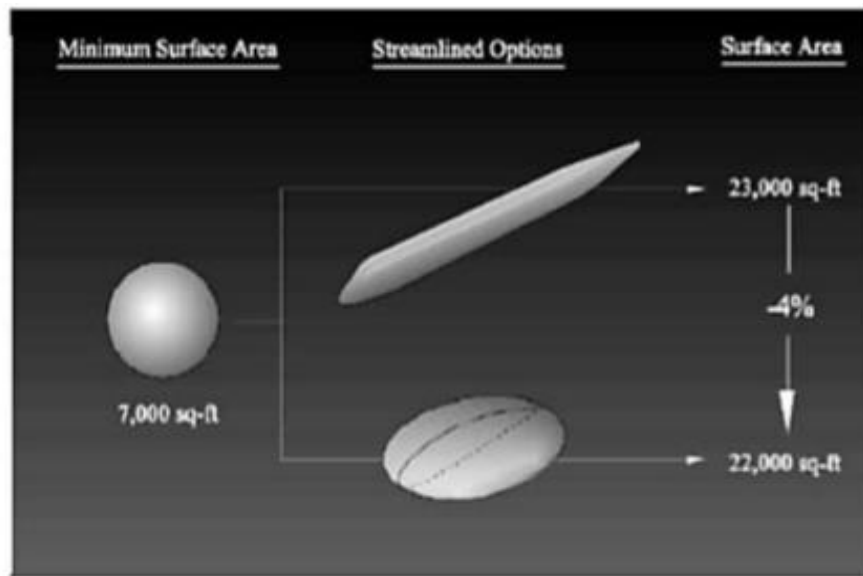


Figure 1.5: Three Conical Forms

Three canonical forms shown in figure 1.5 each sized to hold 800 passengers, were considered. The sphere has minimum surface area however, it is not streamlined options include the conventional cylinder and a disk, both of which are nearly equivalent surface area. Next, each of these fuselages is placed on a wing that has a total surface area of 15,000 ft. Now the effective masking of the wing by the disk fuselage results in a reduction of total aerodynamic wetted area of 7000ft compared to the cylindrical fuselage plus wing geometry as shown in figure 1.6.

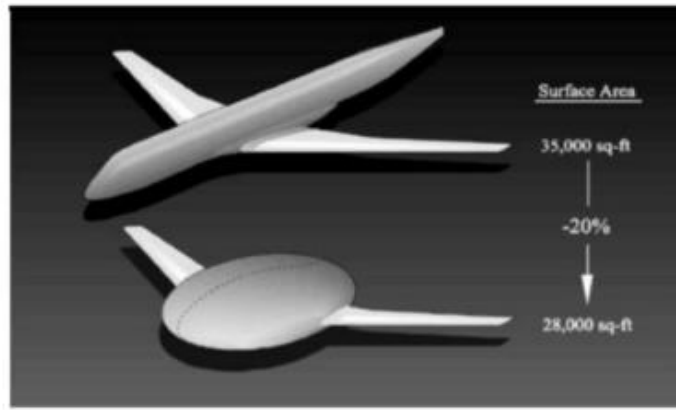


Figure 1.6: Cylindrical Fuselage Wing Geometry

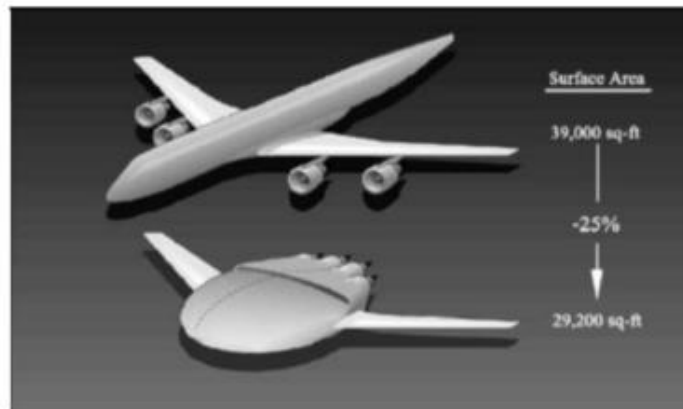


Figure 1.7: Shows the Total Difference in Wetted Area Of 10,200ft

Next, adding engines as shown in figure 1.7, provides a difference in a total wetted area of 10,200 ft. (weight and balance require that the engines be located all on the disk configuration). Finally, adding and required control surfaces to each configuration as shown in fig.1.8 results in a total wetted area difference of 14,300ft or a reduction of 33%. Because the cruise lift-to-drag ratio is related to the wetted area aspect ratio b^2/S_{wetted} the BWB configuration implied a substantial improvement in aerodynamic efficiency.

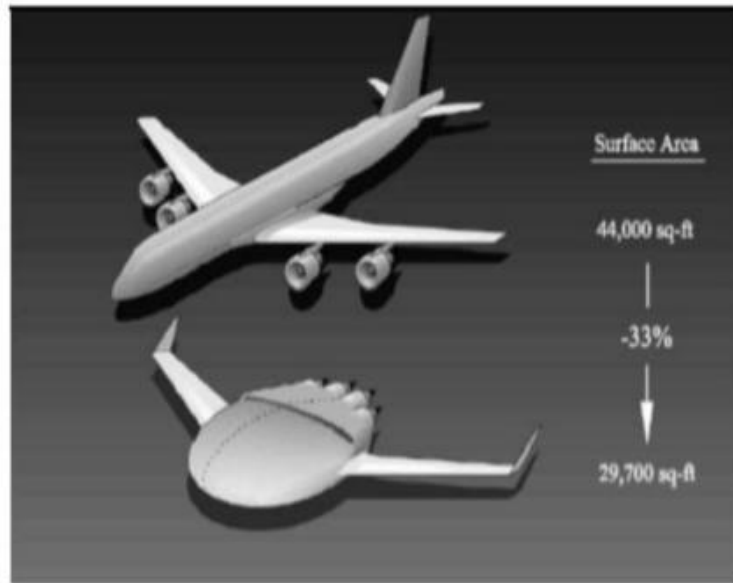


Figure 1.8: Total Wetted Area Difference of 14,300ft

The fuselage is also a wing, an inlet for the engines, and a pitch control surface. Verticals provide the directional stability, control, and act as a winglet to increase the effective aspect ratio, blending and smoothing the disk fuselage into the wing achieved the transformation of the sketch into a realistic airplane configuration.

1.3 Comparison of Aerodynamic, Inertial and Cabin Pressure Loads

The unique element of the BWB structure is the central body as the passenger cabin. It must carry the pressure load bending, and as a wing, it must carry the wing bending load. A comparison of the structural loading of a BWB with that of a conventional configuration is given in Figure 1.9.

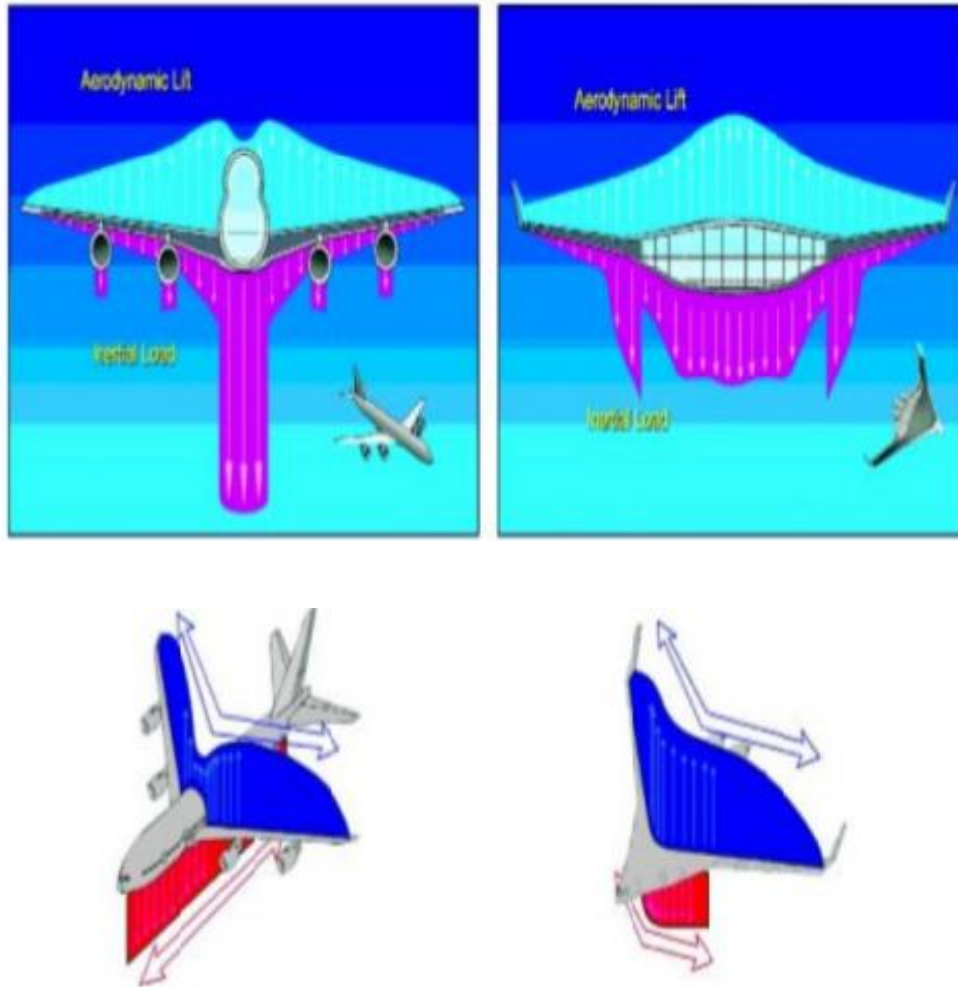


Figure 1.9: Structural Loading of BWB with Conventional Configuration

1.4 Key Concepts of BWB Design

Since the design of the BWB wing in 1988, it has been refined to its current state. The principal concept behind the current iteration of the BWB is the blending of the various components of the plane, including the fuselage, wings, and engines. Into a single lifting surface. As a result, the BWB fuselage is harder to distinguish from the wing (i.e. it is harder to tell where the wings end and the fuselage begins).

There are some key concepts to note about the design of the BWB

1. **The BWB is a tailless aircraft:** Because of the disc-shaped nature of the fuselage. The BWB does not have a tail. As a result, the BWB does not have a rudder.

2. **The engine location of the BWB:** Another important characteristic of the BWB design is the position of the engines which are located at the aft sections of the plane. Because of the weight and balance considerations of the plane, the engines needed to be placed at the rear of the plane, the fuselage can serve as an inlet for the intake of air.
3. **Control surfaces:** The control surfaces of the wing are located along the leading and trailing edges of the winglets. The number of control surfaces varies from 14 to 20 depending on the BWB design.

1.5 Advantages of the Blended Wing Body Aircraft

The BWB has several distinct advantages over conventional tube aircraft. Some of these advantages are outlined below:

1. **Higher fuel efficiency:** Initial testing of the BWB aircraft has indicated that it can here up to a 27% reduction in the fuel burn during flight.
2. **Higher payload capacity:** Due to the blended nature of the fuselage, the fuselage is no longer distributed along the centerline of the aircraft. As a result, the fuselage is more spread out, allowing for greater volume and larger payload capacity.
3. **Lower take-off weight:** early design concepts have determined that the BWB can have up to a 15% reduction of takeoff weight when compared to the conventional baseline.
4. **Lower wetted surface area:** the concept design result in a total wetted difference of 14300 ft², a 33% reduction in wetted surface area. This difference implies a substantial improvement in aerodynamic efficiency.
5. **Commonality:** One of the greatest advantages of BWB is the commonality of the size and application. Firstly, the commonality of the components of the airplane will allow it the payload of the airplane to be varied at little cost. For the 250, 350, and 450- passenger capacity of the BWB, many components are interchangeable. This interchangeability serves to derive sown the cost of the aircraft. Secondly, the commonality of function allows the BWB to be used as a fighter, troop transport, tanker, and stand-off bomber in addition to its function as a commercial airliner.

CHAPTER 2

COMPUTATIONAL FLUID DYNAMICS

2.1 Introduction

Computational fluid dynamics usually abbreviated as CFD is a branch of fluid mechanics that uses numerical methods and algorithm to solve and analyze problems that involve fluid flows or computational fluid dynamics (CFD) is a computer-based tool for simulating the behavior of system involving fluid flow heat transfer and other related physical processes. It works by solving the equation of fluid flow (in a special form) over a region of interest, with specified (known) conditions on the boundary of that region.

Computers are used to perform the calculations required to simulate the interactions of liquid and gases with surfaces defined by boundary conditions. With high-speed supercomputers, a better solution can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experiment and validation coming in full-scale testing e.g. flight tests.

The fundamental basis of almost all CFD problems is the Navier-stokes equation equations which define any single-Phase (gas or liquid, but not both) fluid flow. These equations can be simplified by removing terms describing viscous actions to yield the Euler equation. Further simplification by removing the terms describing vorticity yields the full potential equation. Finally, for small perturbations in subsonic and supersonic flows (not transonic or hypersonic) these equations can be linearized to yield the linearized potential equations.

2.2 Applications of CFD

CFD is used by engineers and scientists in a wide range of fields. Typical application includes: -

1. Process Industry: mixing vessels, Chemical reactors.
2. Building Services: ventilation of buildings, such as atriums.
3. Health and Safety: investigating the effects of fire and smokes.
4. Motor Industry: Combustion modeling, car aerodynamics.

5. Electronics: Heat transfer within around circuit boards.
6. Environmental: Dispersion of pollutants in the air of water.
7. Power and Energy: optimization of the combustion process.
8. Medical: blood flow through the grafted blood vessel.

2.3 CFD Methodology

CFD can be used to determine the performance of a component at the design stage, or it can use to analyze difficulties with an existing component and lead to an improved design. For example, the pressure drop through a component may be considered excessive. The first step is to identify the region of interest. The geometry of the region of interest is then defined. If the geometry already exists in CAD improves directly. The mesh is then created. After importing the mesh into the preprocessor other elements of the simulation including the boundary condition (inlet, outlet, etc.) and fluid properties are defined.

The flow solver is run to produce a file of results that contains the variation of velocity, pressure, and any other variables throughout the region of interest. The geometry of the region of the interest is then defined. If the geometry already exists in CAD improves directly. The mesh is then created. After importing the mesh into the preprocessor other elements of simulation including the boundary conditions (inlet, outlet, etc.) and fluid properties are defined.

The flow solver is run to produce a file of results that contains the variation of velocity, pressure, and any other variables throughout the region of interest. The result can be visualized and can provide the engineer with an understanding of the behavior of the fluid throughout the region of interest. This can lead to design modifications that can be treated by changing the geometry of the CFD model and seeing the effect.

The purpose of performing a single CFD simulation is split into four components.

1. Creating the geometry/mesh
2. Defining the physics of the model.
3. Solving CFD problems.
4. Visualizing the result in the post-processor.

2.3.1 Creating Geometry/Mesh

This interactive process is the first pre-processing stage. The objective is to produce a mesh for input to the physics of the pre-processor. Before a mesh can be produced, a closed geometry solid is required. The geometry and mesh can be created in the meshing applications or any of the geometry mesh creation tools. The basic step involves:

1. Defining the geometry of the region of interest.
2. Creating the region of fluid flow. Solid region and surface boundary names.
3. setting properties for the mesh.

This pre-processing is highly automated. In CFX geometry can be imported from most major CAD Packages using native formats, and the mesh of the control volume is generated automatically.

2.3.2 Defining the physics of the model

This interactive process is the second pre-processing stage and is used to create input required by the solver. The mesh files are located into physics pre-processor CFX-Pre. The physical models that are to be included in the simulation are selected. The fluid properties and boundary conditions are satisfied.

2.3.3 Solving the CFD problems

The component that solves the CFD problems is called the solver. It produces the required result in a non-interactive batch process. A CFD problem is solved as follows:

- a). The partial differential equations are integrated over all the control volumes in the region of interest. This is equivalent to applying a basic conservation law (for example, for mass or momentum) to each control volume.
- b). These integral equations are converted to a system of algebraic equations by generating a set of approximations for the term in the integral equations.
- c). The algebraic equations are solved iteratively.

An iterative approach is required because of the non-linear nature of the equations and as the solution approaches the extra solutions it is said to coverage. Each iteration an error or residual is reported as a measure of the overall conservation of the flow properties. How the final solution does is to extract a solution on several factors including the size and shape of the control volumes and size of the final residual. Complex Physical processes such as combustion and turbulence are often modeled using empirical relationships. The approximation inherent in this model also contributes to the difference between the CFD solutions and the real flow. This solution requires no user interaction and is therefore usually carried out as a batch process. The solver produces a result file that is then passed to the post-processor.

2.3.4 Visualizing the result in the post-processor

The post-processor is the component used to analyze, visualize, and present the resulting interactivity post-processing includes anything from obtaining point values to complex animated sequences.

Example of some important features of post-processors are:

1. Visualization of the geometry and control volumes.
2. Vectors plot showing the direction and the magnitude of the flow.
3. Visualization of the variation of scalar variables (Variable which has only magnitude, not direction, such as temperature pressure and speed) through the domains.
4. Quantitative numerical calculations
5. Animation
6. Charts showing the graphical plots of variables.

2.4 Discretization Methods

The stability of the selected discretization is generally established numerically rather than analytically as with simpler linear problems. Special care must also be taken to ensure that the discretization handles discontinuous solutions gracefully. The Euler equations and Navier-Stokes equations both admit shocks and contact surfaces.

Some of the discretization methods being used are:

1. Finite Volume Method
2. Finite Element Method
3. Spectral Element Method

2.4.1 Finite Volume Method

The finite volume method (FVM) is a common approach used in CFD codes as it has an advantage in memory usage and solution speed especially for large problems High Reynolds number and turbulent flows, and source term dominated flow (like combustion).

In the finite volume method, the governing partial differential equation (typically Navier-Stokes equations and the turbulence equation) are recast in a conservative form, and then solved over discrete control volumes. This discretization guarantees the conservation fluxes through a particular control volume. The finite volume equation yields the governing equation in the form.

$$\frac{\partial}{\partial t} \iiint_V Q \, dV + \oiint_S \mathbf{F} \cdot d\mathbf{S} = 0$$

Where Q is the vector of conserved variables F is the vector of fluxes (see Euler equation and Navier-stokes equation), V is the volume of the control volume element and S is the surface area of the control volume element.

2.4.2 Finite Element Method

The finite difference method (FDM) has historical importance and is simple to program. It is currently only used in a few specialized codes which handle complex geometry with high accuracy and efficiency by using embedded boundaries or overlapping grids (with the solution interpolated across each grid).

$$\frac{\partial Q}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = 0$$

Where Q is the vector of conserved variables F , G and H are the fluxes in the x , y , z direction respectively.

2.4.3 Spectral Element Method

The spectral element method is a finite element type method. It requires the mathematical problem (the partial differential equation) to be cast in a weak formulation.

This is typically multiplying the differential equation by an arbitrary test function and integrating over the whole domain. Purely mathematically, the test functions are completely arbitrary- they belong to infinitely dimensional function space. An infinitely dimensional function space cannot be represented on a discrete spectral mesh. And this is where the spectral element discretization begins. The most crucial thing is the choice of interpolating and test functions. In a standard low order, FEM in 2D. for quadratic element most important choice is the bilinear test or interpolating function of the form

In a spectral method, however the interpolating and the test functions as chosen to be polynomial of a very high order (typically e.g. of the 10th order in CFD applications). This guarantees the rapid convergence of the method. Furthermore, very efficient integration procedures must be used, since the number of integrations to be performed in numerical codes is big. Thus, high order Gauss integration quadrature are employed since they achieve the highest accuracy with the smallest number of computations to be carried out. At the time there are some academic CFD codes based on the spectral element method and some more are currently under development since the new time-stepping schemes arise in the scientific world. You can refer to the C-CFD website to see movies of incompressible flows in channels stimulated with a spectral element solver or to the numerical mechanics' website to see a movie of the Lid-driven cavity flow obtained with a completely novel unconditionally stable time-stepping scheme combined with a spectral element solver.

CHAPTER 3

SOFTWARE PACKAGES USED

3.1 SolidWorks

SolidWorks is a solid modeling computer-aided design (CAD) and computer-aided engineering (CAE) computer program. SolidWorks is published by Dassault Systèmes.

SolidWorks is a solid modeler and utilizes a parametric feature-based approach which was initially developed by PTC (Creo/Pro-Engineer) to create models and assemblies. Parameters refer to constraints whose values determine the shape or geometry of the model or assembly. Parameters can be either numeric parameters, such as line lengths or circle diameters, or geometric parameters, such as tangent, parallel, concentric, horizontal or vertical, etc. Numeric parameters can be associated with each other through the use of relations, which allows them to capture design intent.

The design intent is how the creator of the part wants it to respond to changes and updates. Building a model in SolidWorks usually starts with a 2D sketch. The sketch consists of geometry such as points, lines, arcs, conics, and splines. Dimensions are added to the sketch to define the size and location of the geometry. Relations are used to define attributes such as tangency, parallelism, perpendicularity, and concentricity. The parametric nature of SolidWorks means that the dimensions and relations drive the geometry, not the other way around. The dimensions in the sketch can be controlled independently, or by relationships to other parameters inside or outside the sketch. SolidWorks also includes additional advanced mating features such as gear and cam follower mates, which allow modeled gear assemblies to accurately reproduce the rotational movement of an actual gear train. Finally, drawings can be created either from parts or assemblies.

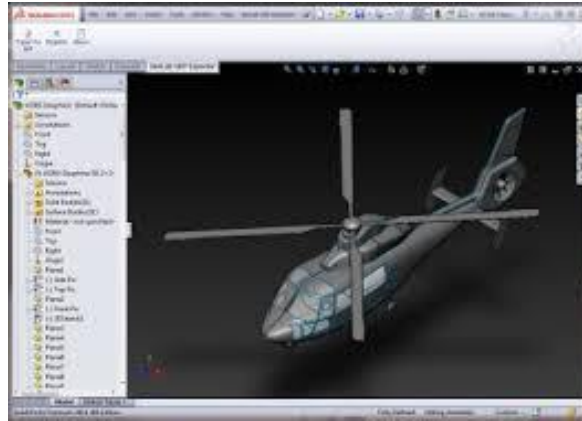


Figure 3.1: Assembly Drawing In SOLIDWORKS

3.1.1 SolidWorks in Aerospace Industry

1. Designers and engineers for next-generation aerospace products and systems must constantly innovate to create extremely complex parts and large assemblies that satisfy customer requirements, maximize performance and meet strict safety standards.
2. SolidWorks product development solutions simplify your design process, accurately evaluate product performance and safety, and help you achieve efficient and cost-effective development from design to manufacturing.
3. SolidWorks has been implemented by a large number of companies within the Aerospace industry, who benefit from the advanced capabilities of this software.

3.1.2 Advantages of SolidWorks

1. The most commonly used software among engineers and designers.
2. Another advantage of using the simulation software is that it is simple to learn how to use it and creating 3D designs is easier which improves productivity.
3. This software gives the designer the ability to import data and translate it, store it securely, and maintain its flexibility and accessibility.
4. Engineers and designers see this software as an innovative way to solve project challenges.
5. Its capability to produce improved designs increases the efficiency of the production and decreases inaccuracies.

3.2 ANSYS

ANSYS is a high-performance general-purpose fluid dynamics program that has been applied to solve wide-ranging fluid flow problems for over 20 years. ANSYS uses advanced solver technology to achieve a reliable and accurate solution quickly and robustly. The modern highly parallelized solver is the foundation for an abundant choice of a physical model to capture virtually any type of phenomenon related to fluid flow.

3.2.1 ANSYS Fluent

ANSYS fluent consists of 4 software modules such as:

1. Design generation software (Design modeler)
2. Mesh generation software
3. Physics pre-solver (Setup)
4. Solver (Solution)
5. Post-processor (Results)

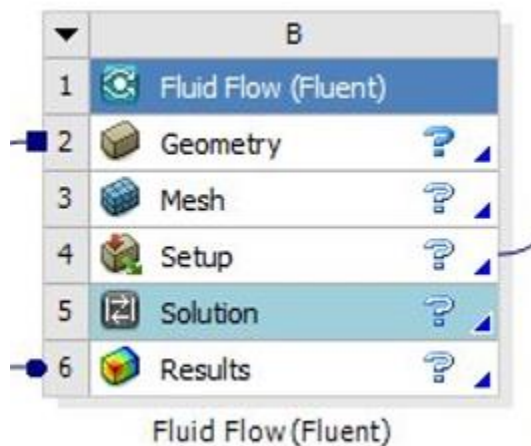


Figure 3.2: ANSYS Fluent

CHAPTER-4

DESIGN PROCESS

4.1 Airfoil Selection

For 2D airfoil selection in the conceptual design phase, a simple and basic approach was adopted as the airfoil details were analyzed using the airfoil investigation database from airfoiltools.com. EPPLER, MARTIN HAPPERLE, and NACA series were analyzed for the BWB conceptual design. The airfoil selection process was focused on the airfoil components to achieve favorable pressure distribution, maximum lift, and minimum drag coefficient.

The Martin Happerle, MH-45 was best suited for our selected geometry which is cambered airfoil and the same airfoil is used for the center body, wing root, and wingtip. The airfoil characteristics are mentioned in Table 1.

Table 1: Characteristics of Airfoil

Parameters	Dimensions	Parameters	Dimensions
Thickness	9.85%C	Low C_m	+0.0145
Chamber	1.7%C	Max C_L Angle	9.5^0
Trailing-Edge Angle	4.4^0	Max L/D	66.664
Lower Flatness	66.6%	Max L/D Angel	6.5^0
Leading Edge Angle	0.7%	Max L/D C_L	0.792
Max C_L	0.888	Stall Angle	6.5^0

MH-45 airfoil gives a comparatively high maximum lift coefficient, can be used till Reynolds number of 100000, and has zero lift angle and has been used successfully in a tailless model airplane. The airfoil has been depicted in figure 4.1.

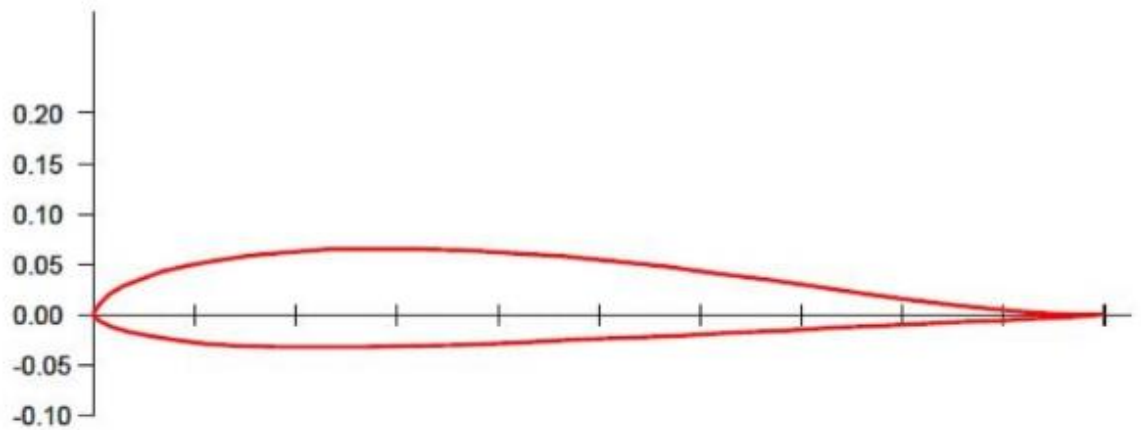


Figure 4.1: MH-45 Airfoil

4.3 Geometric Parameters

Based on the defined scope of the project, we have focused on the geometrical aspects of BWB wing aircraft. By studying many papers, we have selected the BWB UAV geometry from the Design of Blended Wing BODY Unmanned Aerial Vehicle by Jeffrey L. Williams, US. This unique blended wing design includes a wing twist on the outboard of the wing and an inverted 'W' shaped planform to provide lateral and longitudinal stability and smooth and even flight characteristics throughout the range of expected flight envelope. These flight characteristics are crucial to providing a stable reconnaissance planform with favorable stall speeds, an increased payload, and the ability to and launch without the danger of explosion.

In this wing, the assembly comprises a central main wing and outer external wing smoothly blended from root to tip. The airfoil has a Reynolds number of 63,000. Geometric details of the design are mentioned in Table 2.

Table 2:Geometric Details of Design

Parameters	Dimensions	Parameters	Dimensions
Centre Chord	0.89 m	Half Span	0.86 m
Root Chord	0.42 m	Sweep Angle	30 ⁰
Tip Chord	0.26 m	Dihedral Angle	0 ⁰
Twist Angle	0 ⁰	Surface Area	0.714 m ²

4.4 Design Process

The design is further developed in SolidWorks by the following steps:

STEP 1: Importing of an airfoil coordinates in SolidWorks

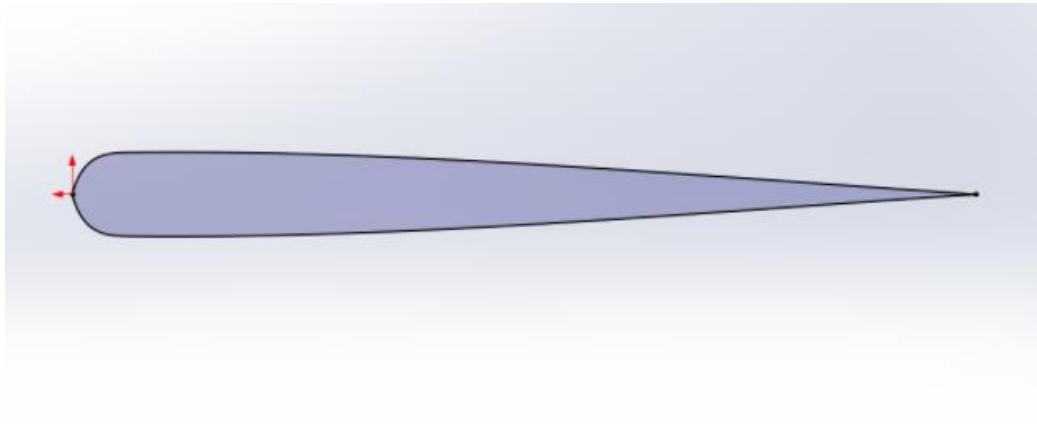


Figure 4.2 Importing airfoil in SolidWorks

STEP 2: Resizing and projection of airfoil in different planes according to dimensions

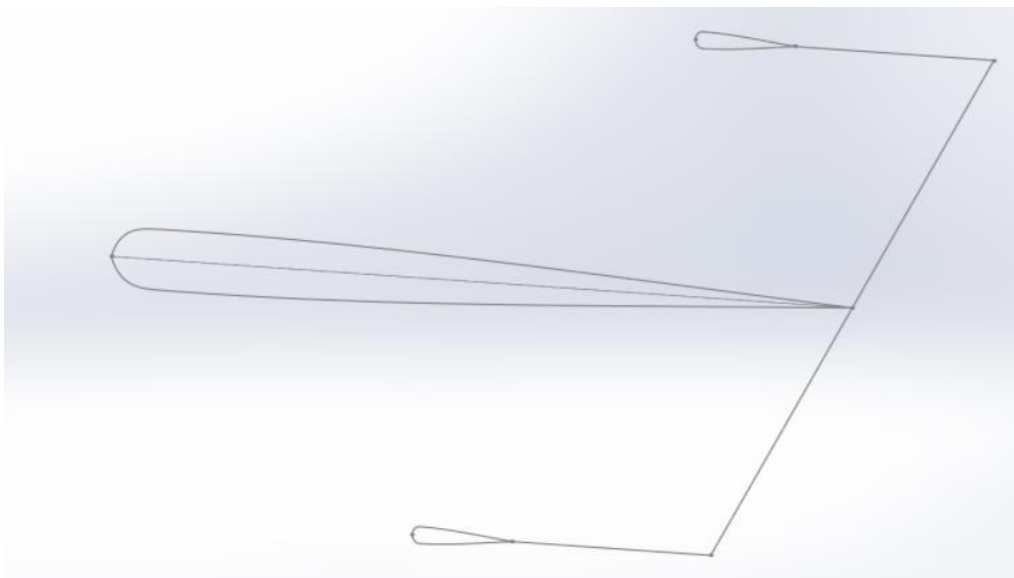


Figure 4.3 Resizing And Projection On Different Planes

STEP 3: Creating a projection of wingtip in the perpendicular plane to create winglets

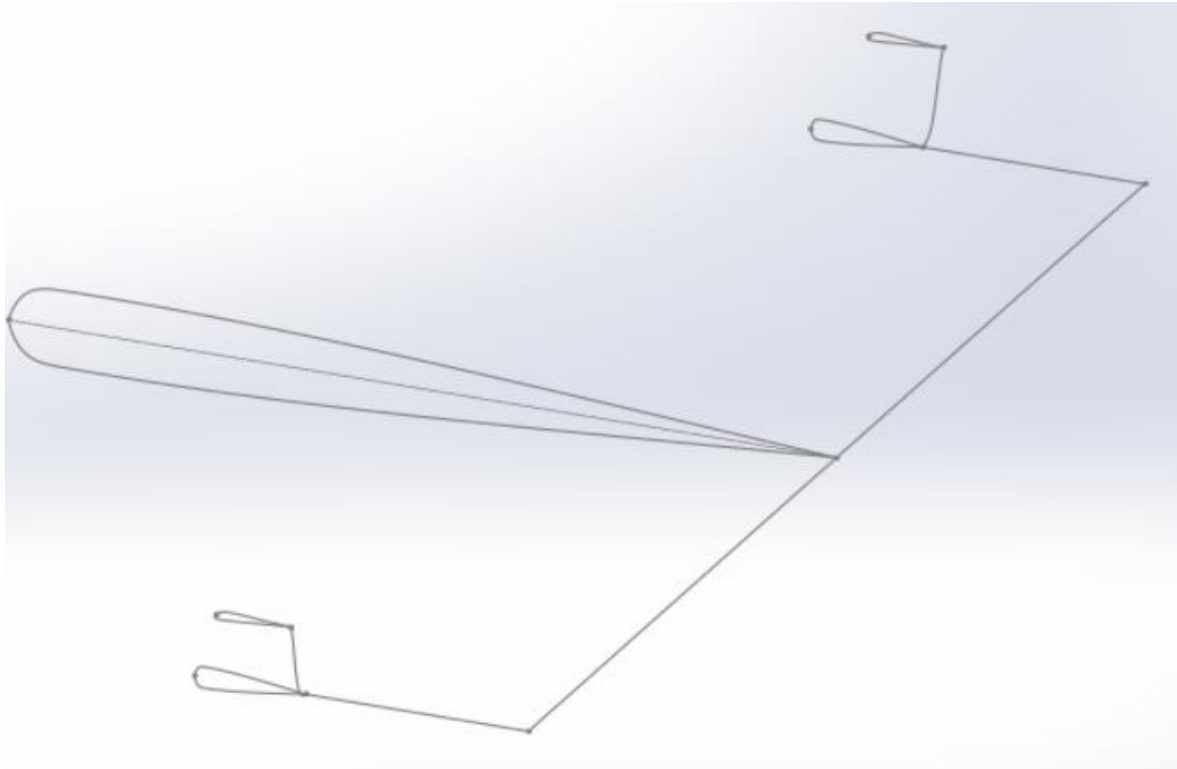
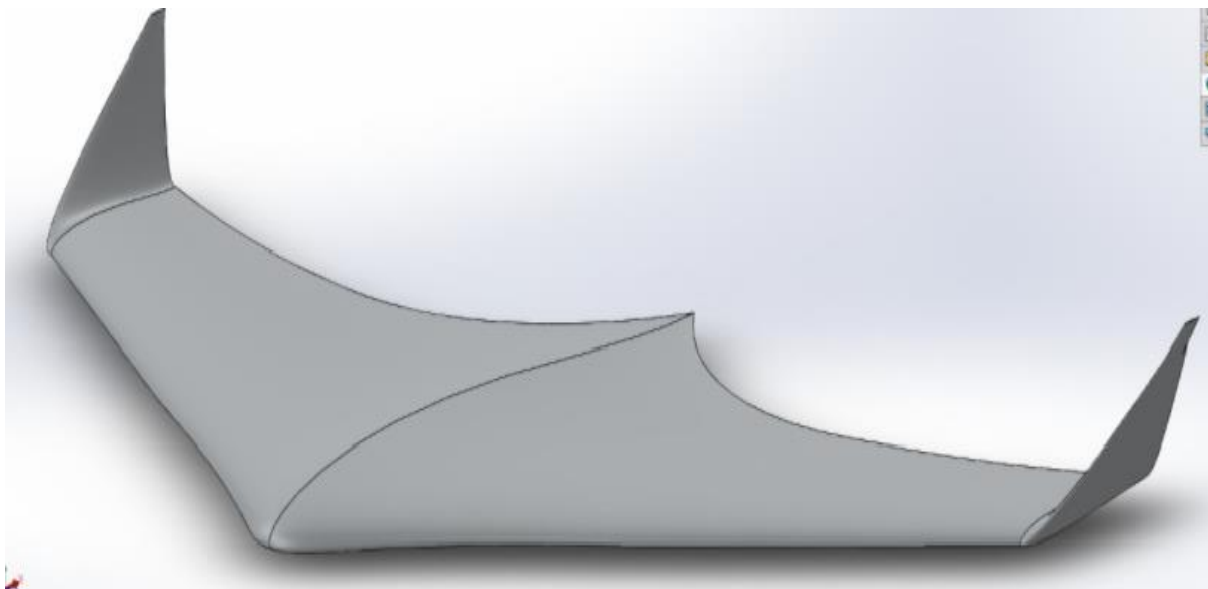
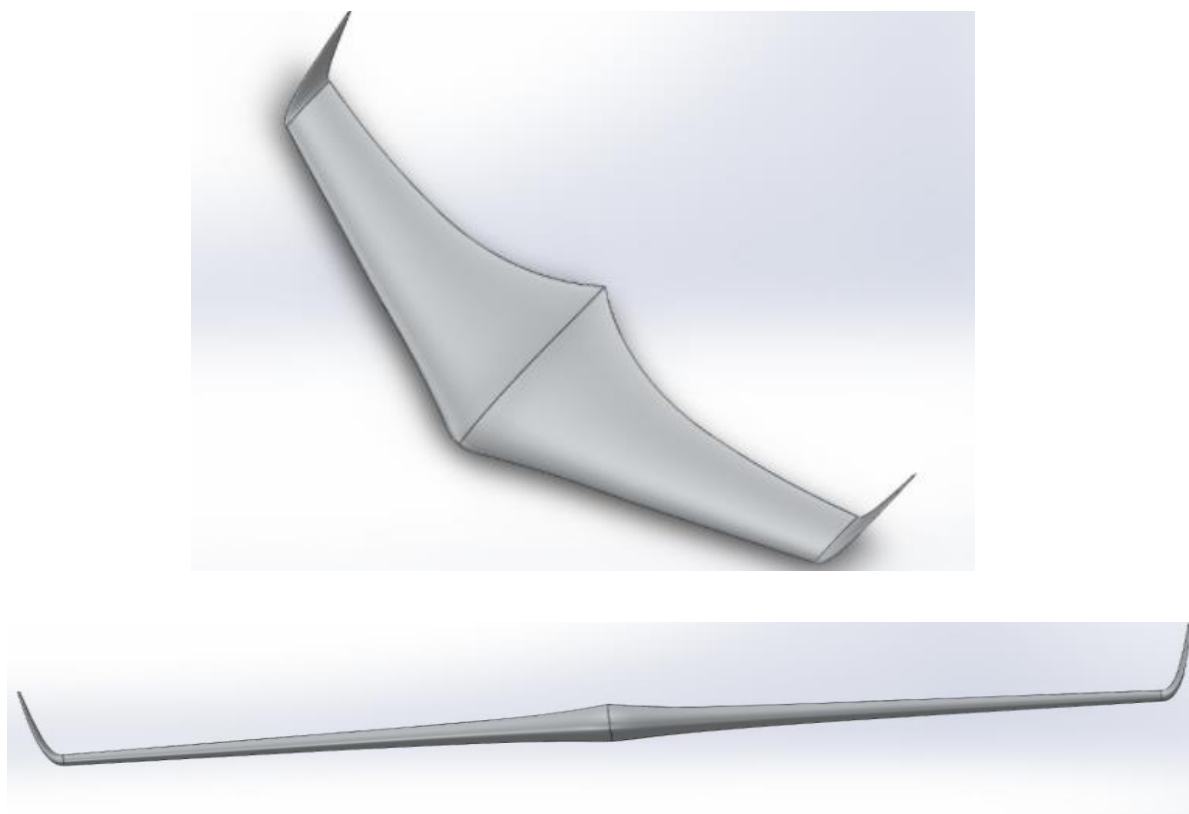


Figure 4.4 Projection for Fins

STEP 4: Creating loft (BLEND) between these surfaces to get the final model ready





Figures 4.5: Different Views of Complete BWB Design

CHAPTER-5 MESHING

5.1 ANSYS Design Modeler and Meshing

ANSYS design modeler and meshing is a popular proprietary software package used for CAD and mesh generation. Some open-source software includes Open FOAM and open FVM etc. The present discussion applies to ANSYS Design modeler and meshing CFD software. It can create structured, unstructured, multi-block, and hybrid grids with different cell geometries.

5.1.1 Geometric Modelling

ANSYS design modeler and meshing are also meant to mesh a geometry already created in other CAD Software. Therefore, geometric modeling features primarily meant to clean-up an imported CAD model. Never-the-less there are some very powerful geometry creation, editing, and repair tools available in ANSYS design modeler which assist in arriving at the meshing stage quickly.

Apart from the regular points, curves, surface creation, and editing tools, ANSYS meshing especially can do Build topology which removes unwanted surfaces, and then you can view if there are any holes in the region of interest for meshing. Holes are typically identified through the color of curves. The following color code is used by ICEM MESHING:

1. **Yellow:** Curve attached to a single surface- possibly a hole exists.
2. **Red:** Curve shared by two surfaces.
3. **Blue:** curve shared by more than two surfaces.
4. **Green:** Unattached curve is not attached to any surface.

5.1.2 Creation of a Fluid Domain

After the defeaturing of the imported geometry is completed, an enclosure is created specifying the fluid flow domain. The enclosure is such that fully encloses the body and has an extra margin of 10%.

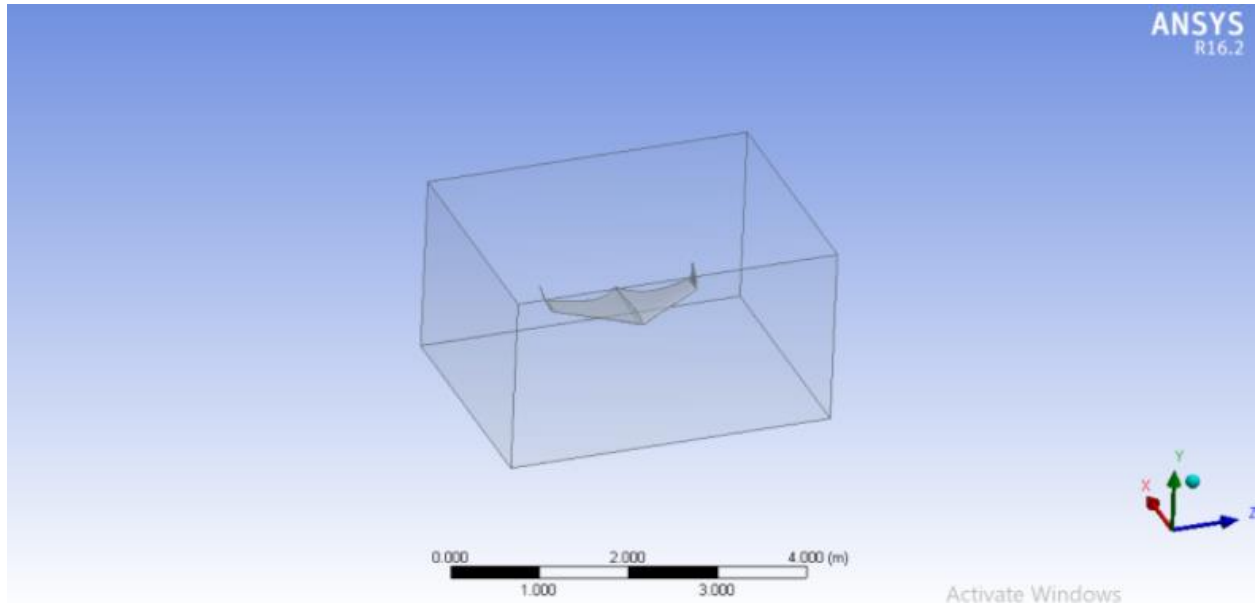


Figure 5.1 Fluid Domain

5.2 Meshing

There are often misunderstandings regarding structured/unstructured mesh, meshing algorithm, and solver. A mesh may look like structured but may/may not have been created using a structured algorithm-based tool. Our ANSYS FLUENT is an unstructured solver which can only work on an unstructured mesh, even if we provide it with a structured looking mesh created using a structured/unstructured algorithm based meshing tool.

5.2.1 Mesh Process

ANSYS mesh process consists of 5 basic steps:

1. Select Physics preference in ANSYS Meshing (CFD).
2. Select the meshing method (AUTOMATIC)
3. Insert Local mesh setting.
4. Preview and generate the mesh.
5. Check mesh quality.

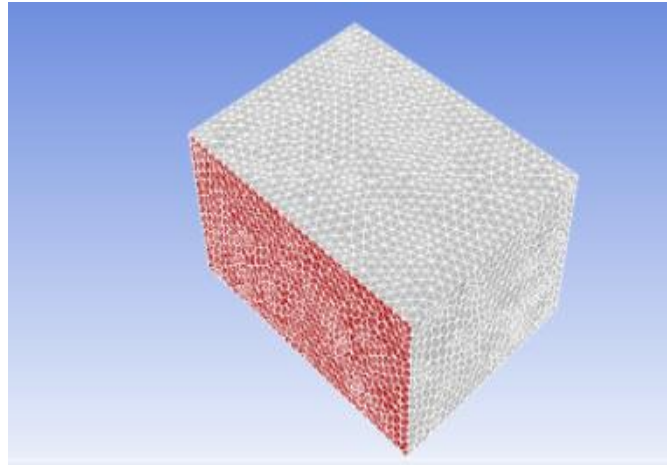


Figure 5.2: Surrounding Meshing

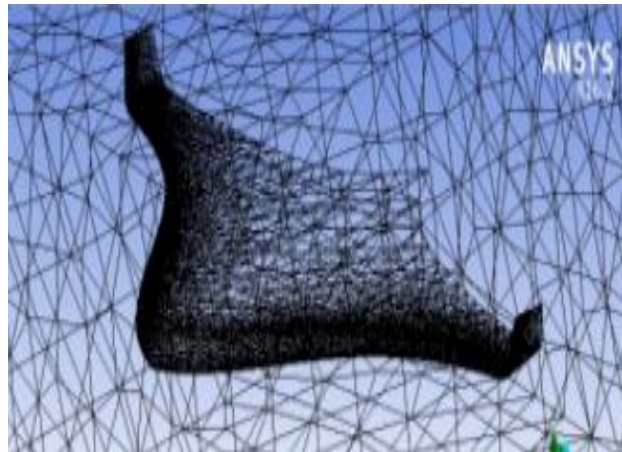


Figure 5.3: BWB Meshing

5.2.2 Inserting Named Section

Select the appropriate faces and name them as inlet velocity, outlet pressure, and adiabatic walls. These named sections will help in directing the flow and calculation initialization. Figure 5.4 shows the inlet, outlet, and adiabatic walls respectively.

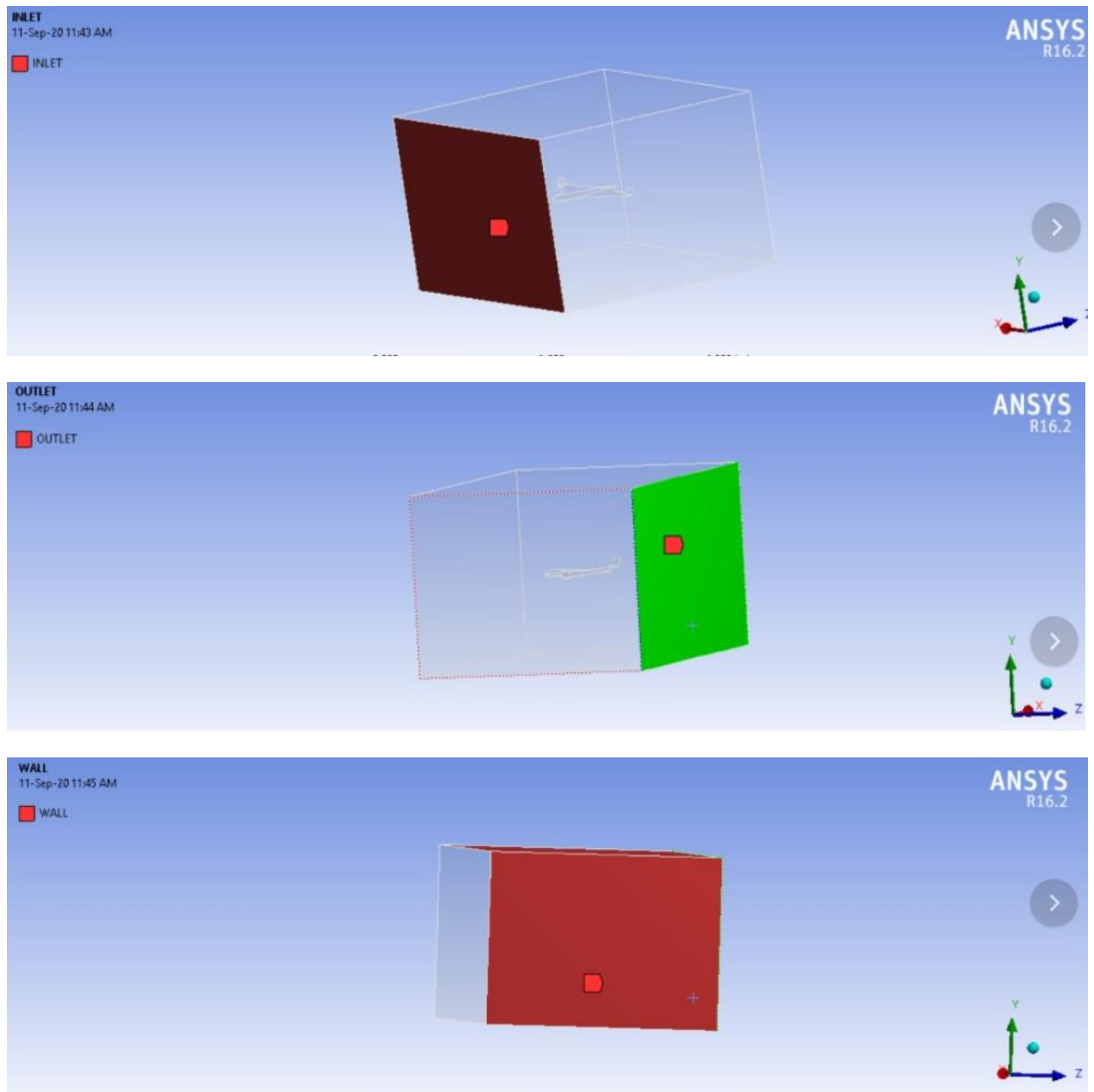


Figure 5.4: Inlet, Outlet, and Wall.

CHAPTER-6

RESULTS

The design is simulated in ANSYS Fluent of Workbench and analysis is carried out at 50 m/s (0.14 Mach). The Reynolds number is kept constant at 63000. The K-epsilon equation is used to solve the above problem because of it being a turbulent flow problem. The convergence criteria of 10^{-3} to 10^{-4} was achieved after 180 iterations. Therefore, the solution is processed for 200 iterations.

Initially, the values obtained are highly dynamic and vary very fast but gradually with a greater number of iterations, the solution converges and we get a constant value of the coefficients. Further, in pressure distribution, the stagnation point appears close to the nose of the aircraft which also validates our simulation.

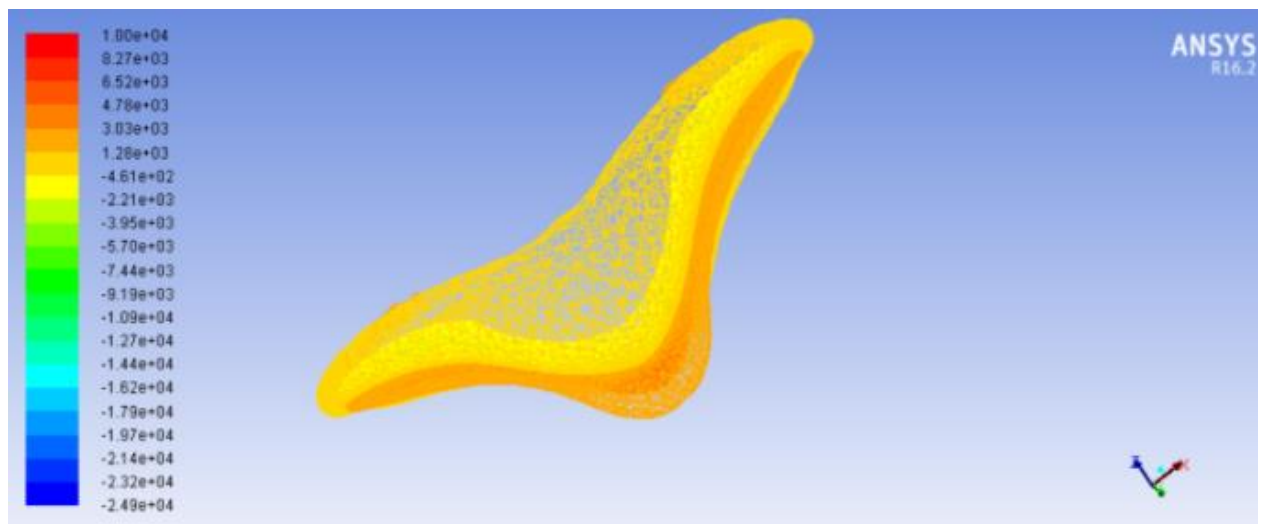


Figure 6.1: Pressure Distribution Over BWB

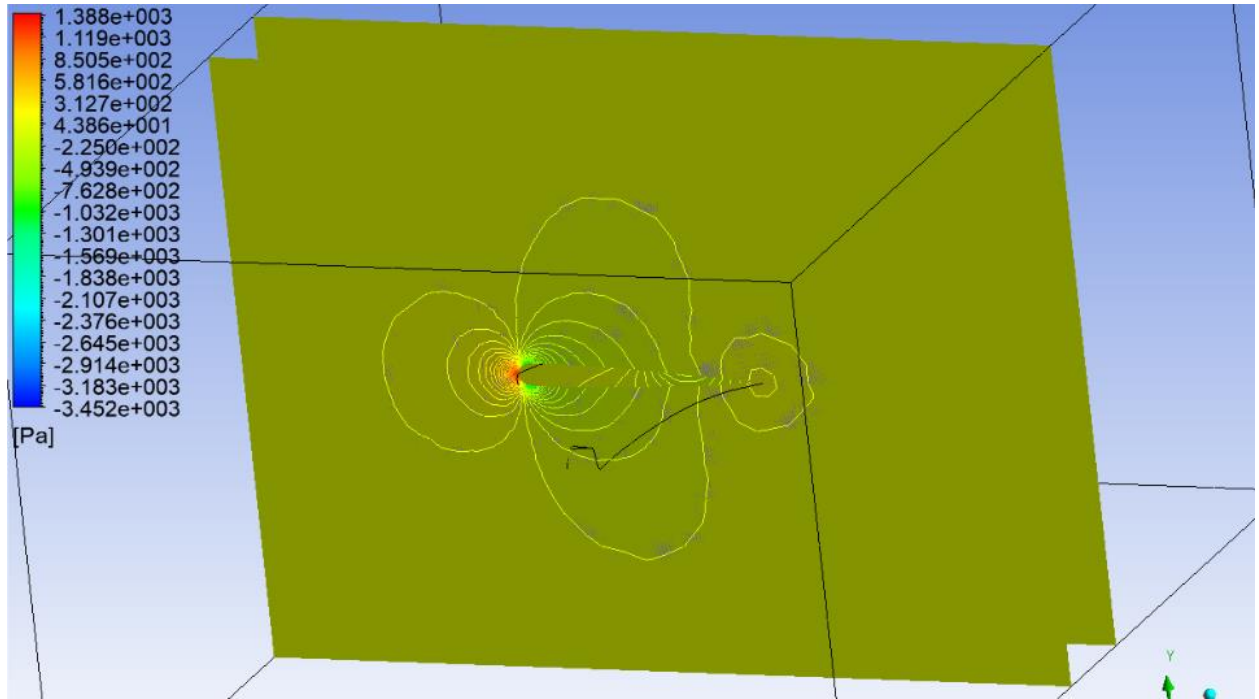


Figure 6.2: Static Pressure Over Center Plane

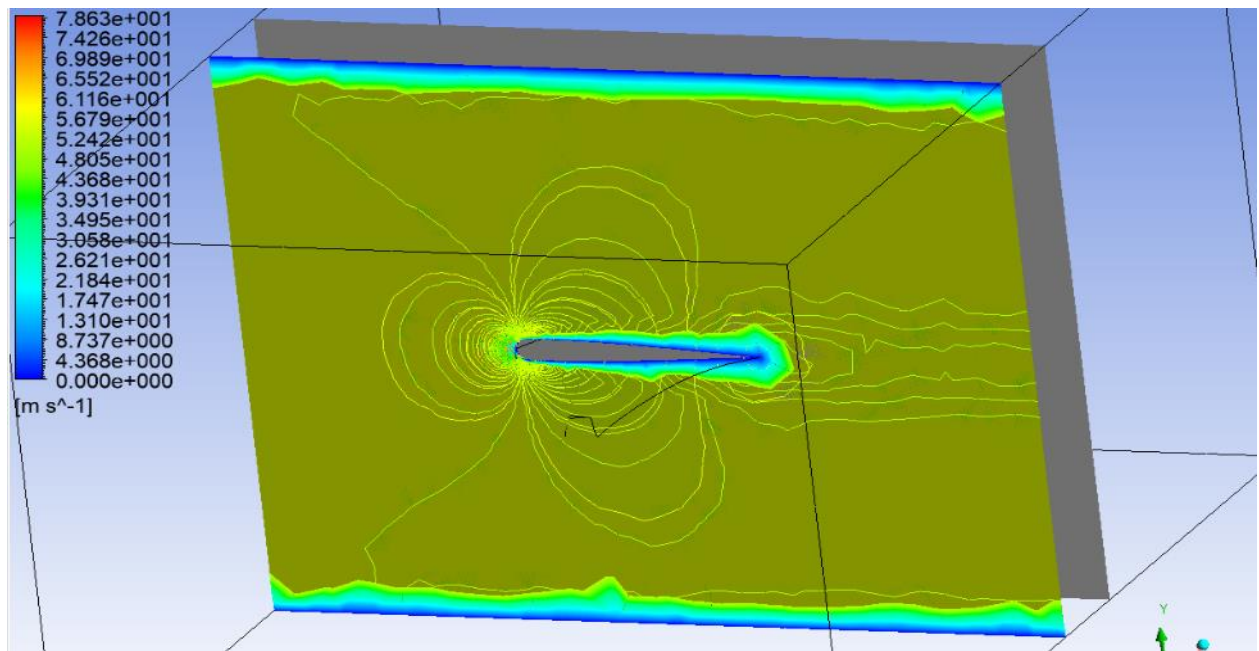


Figure 6.3: Velocity Distribution Over Center Plane

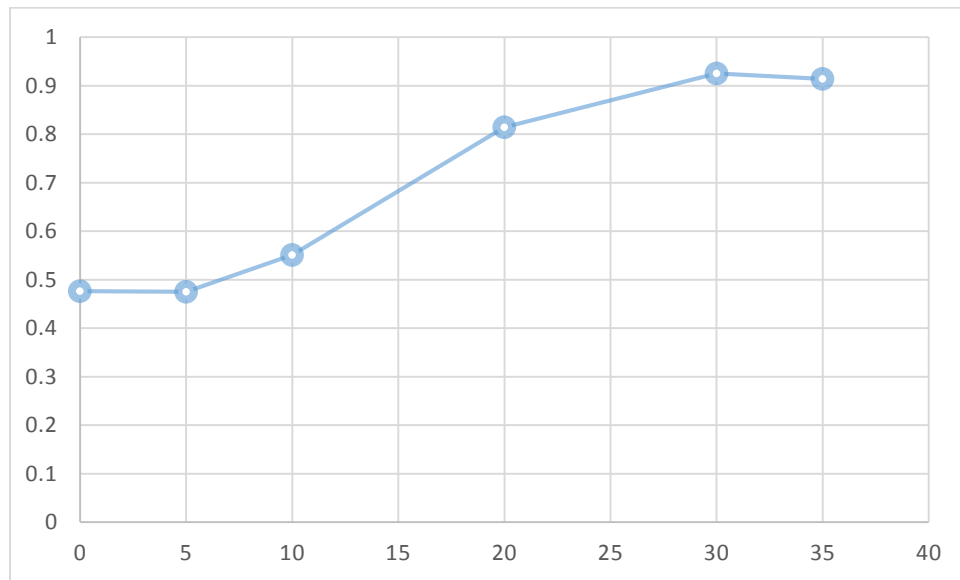
6.2 Force Coefficients

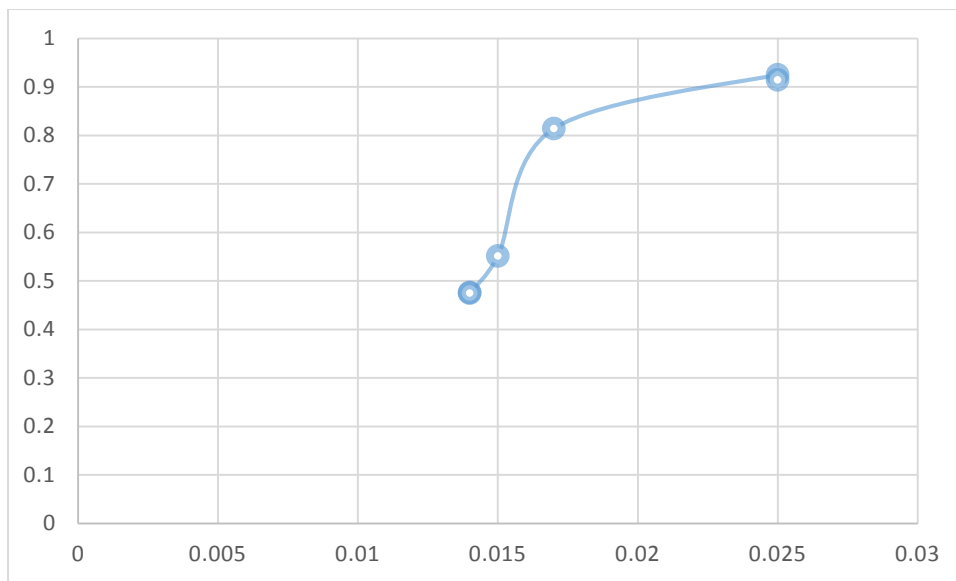
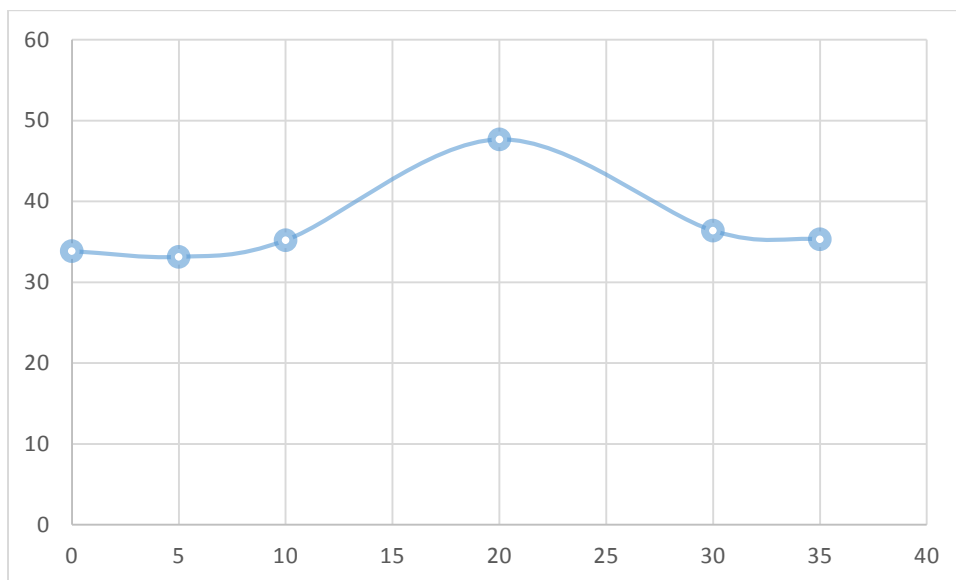
By using ANSYS Fluent, the model is simulated at a different angle of attacks (AOA) keeping the velocity and other parameters as constant. The result obtained after the simulation are mentioned in Table 3.

Table 3: Force coefficients

AoA	Lift Force	Drag Force	L/D	q	S	C_L	C_D	C_L/C_D
0	521.3	15.4	33.85	1531.25	0.714	0.476	0.014	33.85
5	520.3	15.7	33.14	1531.25	0.714	0.475	0.014	33.14
10	603.0	17.1	35.20	1531.25	0.714	0.551	0.015	35.20
20	890.0	18.66	47.68	1531.25	0.714	0.814	0.017	47.68
30	1012.1	27.81	36.38	1531.25	0.714	0.925	0.025	36.38
35	999.3	27.96	35.31	1531.25	0.714	0.914	0.025	35.31

Now, the plot of C_L vs alpha, DRAG POLAR, and C_L/C_D vs alpha is generated.

Figure 6.7: Variation of C_L with Alpha

Figure 6.8: Variation of C_L with C_D Figure 4.6: Variation of L/D with α

CHAPTER-7 CONCLUSION

The lift-to-drag ratio (L/D) is a very significant parameter to pay attention to during the design process. It is the amount of lift generated by a wing or vehicle, divided by the aerodynamic drag it creates by moving through the air. A higher or more favorable L/D ratio is typically one of the major goals in aircraft design; since a particular aircraft's required lift is set by its weight, delivering that lift with lower drag leads directly to better fuel economy in aircraft, climb performance, and glide ratio.

This can be explained in another way. During straight and level flight conditions, the four main forces acting on the airplane are in equilibrium, i.e. lift, weight, drag, and thrust. A higher lift means that more weight is allowable, which in turn means a higher payload capacity (which is very important for transport aircraft). A lower drag value means less thrust will be required and hence lower fuel consumption.

Table 4:L/D of Various Aircrafts.

AIRCRAFT	L/D ratio
Boeing-747	17.7
Lockheed-U-2	25.6
Airbus A-320	19.2
BWB Design	33.85

The above table details the lift-to-drag ratio of various aircraft at cruise conditions. The Boeing 747 provides a better comparison. It is a high-capacity, wide-body commercial airliner and has an L/D ratio of 17.7 The Blended Wing Body configurations has achieved almost double that value.

Through the analysis, it has been found that the blended wing body can obtain better efficiency and high performance at subsonic speeds. The lift generated by the blended wing body was found to be more than the conventional aircraft. There

was also a reduction in drag for the BWB. One of the most significant differences between conventional aircraft and the BWB is that the body or the fuselage of the BWB generates lift which was confirmed by the analysis of the center body.

REFERENCES

[1] “Blended Wing Body Aircraft”

https://en.wikipedia.org/wiki/Blended_wing_body

[2] “Blended Wing Body Design”

<https://www.nasa.gov/centers/langley/news/factsheets/FS-2003-11-81-LaRC.html>

[3] R. Devaraju, R. Naveen Kumar, J. Naresh, Computational Aerodynamic Analysis of Blended Wing Body Aircraft, International Research Journal of Engineering and Technology, Volume: 02, Issue:05, 2015.

[4] Michael Farrow, Wing Embedded Engines for Large Blended Wing Body Aircraft, A Computational Investigation, University of Surrey, 2009.

[5] Zhoujie Lyu and Joaquim R. R. A. Martins, Aerodynamic Design Optimization Studies of a Blended-Wing-Body Aircraft, Journal of Aircraft, 2013.

[6] R.H. Liebeck, Design of the Blended Wing Body Subsonic Transport, Journal of Aircraft, Vol. 41, No.1, 2004.

[7] Spiridon Souris and Ning Qin, Study of the effects of wing sweep on the aerodynamic performance of a blended wing body aircraft, Proceedings of the Institution of Mechanical Engineers Part G Journal of Aerospace Engineering, 2007.

[8] Hassan Muneel Syed, M. Saqib Hameed, and Irfan A. Manarvi, A Review of Swept and Blended Wing Body Performance Utilizing Experimental, FE, and Aerodynamic Techniques, International Journal of Recent Research and Applied Studies, 2011.

[9] Tung Wan and Hei Yang, Aerodynamic Performance Investigation of A Modern Blended-Wing-Body Aircraft Under the Influence of Heavy Rain Condition, International Congress of the Aeronautical Sciences, 2010.

- [10] K. Simhachalam Naidu, G. ShruthiKeerthi, V.V.V Nikhil Bharadwaj, CFD Analysis of Blended Wing Body and B2 Wing, International Journal of Engineering Sciences & Research Technology, Thomson Reuters Endnote, 2016
- [11] Kai Lehmkuehler, KC Wong, and Dries Verstraete, Design and Test of a UAV Blended Wing Body Configuration, International Congress of the Aeronautical Sciences, 2012.
- [12] Caroline Delbert; "Will We One Day Fly-in This 'Blended Wing' Airplane? Airbus Built a Prototype To Find Out", Popular Mechanics, 13 February 2020. (Retrieved 18 February 2020)
- [13] Bradley K (2004) A sizing methodology for the conceptual design of blended-wing-body transports. NASA/CR-2004-213016, September
- [14] Liebeck RH (2002) Design of the blended-wing-body subsonic transport. In: 40th AIAA Aerospace sciences meeting & exhibit, Reno, AIAA-2002-0002.
- [15] Mukhopadhyay V (2005) Blended-wing-body (BWB) fuselage structural design for weight reduction. In: 46th AIAA/ASME/ ASCE/ AHS/ ASC Structures, structural dynamics and materials conference, Austin, AIAA 2005-2349.
- [16] Österheld C, Heinze W, Horst P (2001) Preliminary design of a blended wing body configuration using the design tool Prado. In: Proceedings of the CEAS conference on multidisciplinary aircraft design and optimization, Cologne.
- [17] N. Qin, A. Lavalley, A. Le Moigne, M. Laban, K. Hackett, and P. Weinerfelt, "Aerodynamic Considerations of Blended Wing Body Aircraft", Progress in Aerospace Sciences, 40 (6), 321–343
- [18] H. J. Shim and S. O. Park, "Low-Speed Wind Tunnel Test Results of a BWB-UCAV Model", Procedia Engineering, 67, 50–58 (2013)
- [19] P. Okonkwo and H. Smith, "Review of Evolving Trends in Blended Wing Body Aircraft Design", Progress in Aerospace Sciences, 82, 1–23 (2016).
- [20] Warwick, Graham (Jan 12, 2013). "Hear This - The BWB is Quiet!". Aviation Week.