Performance Test of the Low-Subsonic Wind Tunnel Present at SU Campus

A Thesis

by

Dewan Emran Mahamud Shuvo
ID: ME2103025161
Jiban Ahammed
ID: ME2103025160
Mehedi Hasan Paran
ID: ME2103025162
Sumi Akter
ID: ME2103025168
Anikul Islam
ID: ME2002021017
Saniat Fatema
ID: ME2002021310
Reg. No.: 1205, Session: 2021-2022

Supervisor:

Md. Ahatashamul Haque Khan Shuvo
Assistant Professor
Department of Mechanical Engineering

Submitted to the
DEPARTMENT OF MECHANICAL ENGINEERING
SONARGAON UNIVERSITY (SU)
In partial fulfillment of the requirements for the award of the degree
of
Bachelor of Science in Mechanical Engineering

May 2025

Acknowledgement

First, we pay our gratitude to the Almighty Allah for giving us the ability to work hard successfully. Words actually will never be enough to express our gratefulness.

We would like to express our gratitude and respect to our beloved parents for their love, affection, patient and immense support. We are very grateful to them for their faith in us and with the blessings and encouragement of our parents, we were able to carry out the research work.

We consider it an honor to work with our respected supervisor **Md. Ahatashamul Haque Khan Shuvo** with his guidance, motivation, inspiration and above all for his extreme support and co-operating attitude our thesis work has come to this present form.

We owe to the Department of Mechanical Engineering, Sonargaon University Bangladesh and all the faculty members for their invaluable support, kindness and inspiration in every situation and needs.

Authors

Abstract

The wind tunnel is a tool used in aerodynamic research to study the effects of wind moving over a solid object. This thesis work is done to performance test of a low-subsonic wind tunnel by our Sonargaon university lab. We get Mach number 0.02 and the Reynolds Number is 53578 for a designed test airfoil of 150 mm for a test section of 910 mm of length. With the target of having a flow speed about 7.02 m/s at the test section of 0.32 m^2 cross-sectional area all the other sections of the wind tunnel have been designed and using a sixth-order polynomial equation. The wind tunnel operates at speeds below Mach 0.3 and using Java foil software for calculation.

Table of Contents

Acknowledgement..... I
Abstract..... II
Table of Contents…….. II
List of Table V
List of Figures…….. V
CHAPTER I Introduction ……..……..... 1-5
1.1. Introduction 1
1.2. Wind Tunnel 2
1.2.1. General Purpose Wind Tunnels 3
1.3. History and Background 4
1.3.1. Literature review 5
1.4. Objectives……. 5
CHAPTER II Wind Tunnel Components..... 6-14
2.1 Wind Tunnel Components 6
2.1.1Contraction Cone 6
2.1.2Diffuser 6
2.1.3 Test Section 7
2.1.4. Settling Chamber 7
2.1.5. Honeycomb 8
2.1 6. Screen 8
2.1.7. Turbo Machinery 9
2.1.8. Axial Flow Fan Types 9
2.1.9 Propeller Fans 9
2.10 Tube Axial Fans 10
2.1.11 Vane Axial Fans 10
2.2 Load Transmission..... 11
2.3. Balance 11
2.4 Test object 12
2.5 Sensor 14
2.5.1Digital Anemometer 14

CHAPTER III Airfoil Model, Drag Force & Lift Force..... 15-19
3.1. Air Foil Model…… 15
3.2. Drag Force…… 17
3.3. Drag Force Coefficient……..... 17
3.4. Lift Force…..……..... 18
3.5. Lift Force Coefficient…..…… 18
CHAPTER IV Calculation……..... 20-27
4.1. Flow parameter Calculation 20
4.1.1. Reynolds Number 20
4.1.2. Mach Number 20
4.2 Sample Calculation……..... 21
4.3 Result Analysis 27
CHAPTER V Conclusion……..…….... 28-28
5.1. Conclusion 28
REFERENCES 29

List of Tables

Table 1.1: Comparison of Open Circuit and Closed-Circuit Wind Tunnel……..... 3
Table 4.1: Different Flow Region Based on Mach Number ..……..... .21
Table 4.2: Data Generated by Java Foil for NACA0024 Model ……..…… 21
Table 4.3: Drag force and Lift force with Coefficient…….……... .23

List of Figures

Figure 1.1: Schematic Diagram Showing Different Parts of Wind Tunnel 1
Figure 1.2: Schematic View of a Open-Circuit Wind Tunnel 2
Figure 1.3: Schematic View of a Closed-Circuit Wind Tunnel 2
Figure 2.1: Contraction Cone 6
Figure 2.2: Diffuser Geometry and Notation 7
Figure 2.3: Test Section 7
Figure 2.4: Settling Chamber 8
Figure 2.5: Honeycomb 8
Figure 2.6: Screen and Mesh 9
Figure 2.7: Direct Drive and Belt Driven Propeller Fan 10
Figure 2.8: Direct Drive and Belt Drive Tube Axial Fan. 10
Figure 2.9: Direct Drive vane Axial Fan……. 11
Figure 2.10: Balance Bar……..... 12
Figure 2.11: Schematic Diagram of Test object 13
Figure 2.12: Patterns of Flow Around Airfoil…..... . 13
Figure 2.13: Digital Anemometer…..... . 14
Figure 3.1: Basic Nomenclature of an Airfoil 15
Figure 3.2: Relation between Drag and Lift Force……..……...….. 17
Figure 4.1: Graph of Velocity Distributions……..... 22
Figure 4.2: Graph of Drag force & drag coefficient …….... 23
Figure 4.3: Graph of lift force & Lift coefficient …….... 24
Figure 4.4: Graph of Drag force VS Lift Force ……..…..... 24
Figure 4.5: Graph of Drag force Coefficient VS Lift Force Coefficient……..... .25

CHAPTER I

INTRODUCTION

1.1. Introduction

The wind tunnel is a tool to study fluid flows around a body and the forces generated by the fluid-structure interaction. Using such tool, it is possible to measure global and local flow velocities, as well as pressure and temperature around the body. A wind tunnel experiment provides the velocity values on the model and the flow visualization. It is a critical instrument in the quick and thorough design process of anything that involves fluid dynamics. Based on the flow through the tunnel circuit, the wind tunnel is of two types, open type and closed type. In open type, ambient air enters from one side and exits to the atmosphere after flowing through the tunnel. In closed type wind tunnel, a constant volume of air is allowed to pass through the tunnel circuit continuously Figure 1.1.

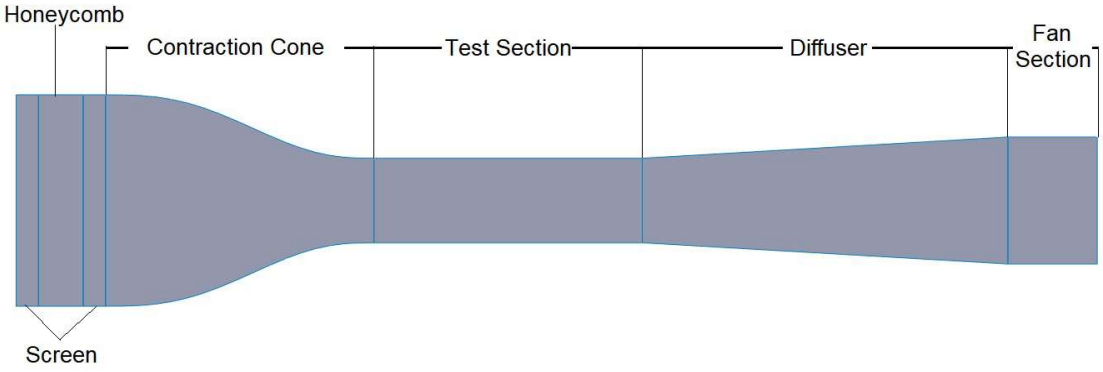


Figure 1.1: Schematic Diagram Showing Different Parts of Wind Tunnel

The main goal of wind tunnel design is to have uniform flow within the test chamber. Test chamber dimension depends on the type of test to be performed and the size of the model to be tested. The wind tunnel's overall dimensions are key factors in its construction and running costs. The trade-off is required between the opposing needs to get a suitable design of the wind tunnel, satisfying the test requirements, space and budget constraints. The design process starts by defining the test chamber dimensions and shape, after that the rest part of the wind tunnel is designed by keeping in view the dimensions of the test chamber. Principal parts of an open type wind tunnel are the test section, contraction, diffuser, settling chamber.

1.2. Wind Tunnel

Wind tunnels are large tubes with air blowing through them. The tunnels are used to replicate the actions of an object flying through the air or moving along the ground. Researchers use wind tunnels to learn more about how an aircraft will fly.
There are many ways to classify the wind tunnels. But two major classifications are due to the circuit type and test section type. The circuit can be either closed circuit and open circuit.

In an open circuit wind tunnel, the air is taken from outside environment and discharged to outside environment again. The air generally follows a straight path from the entrance to the exit of the wind.

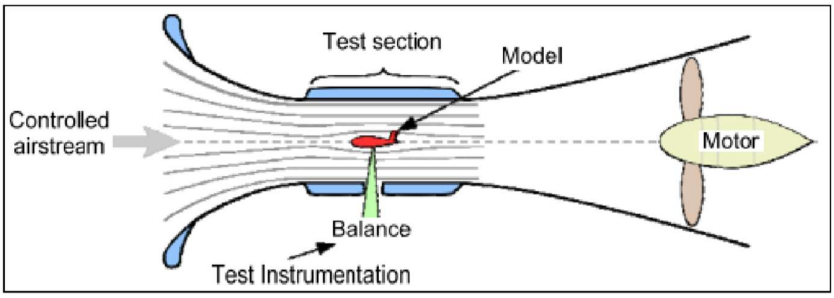


Figure 1.2: Schematic View of a Closed-Circuit Wind Tunnel

In closed circuit wind tunnels, the air is circulated by the help of a power unit continuously. Sometimes, a small amount of air is exchanged with the 5 environments to increase the air quality and have some temperature control.
The test section of the wind tunnels can be either open jet (no solid boundaries around the test section) or closed jet (solid boundaries around the test section). In some wind tunnels, the test sections also have slotted walls.

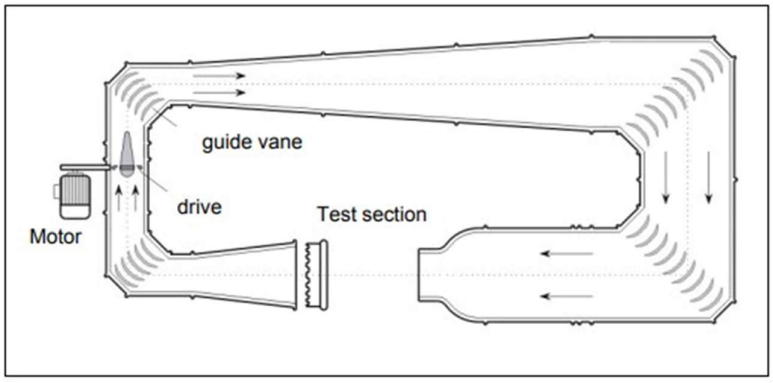


Figure 1.3: Schematic View of a Closed-Circuit Wind Tunnel

Open circuit and closed-circuit wind tunnels have some major advantages and disadvantages. A comparison of closed-circuit wind tunnels and open circuit wind tunnels is given in Table 1.1.

Table 1. 1: Comparison of Open Circuit and Closed-Circuit Wind Tunnels.

Features	Advantages	Disadvantages
Open Circuit Wind Tunnels	Less construction cost	Flow quality is easily affected from wind or room size
	No purging problem in flow visualization	The tunnel requires more energy (for a given size and speed)
		The open circuit tunnels are more
	Less noisy	noisy
Closed Circuit Wind Tunnels	The quality of the flow can be well controlled and independent of atmosphere	Initial investment is higher
	The tunnel requires less energy (for a given size and speed)	Precautions must be taken if used for flow visualization
		Needs cooling if used extensively

1.2.1. General Purpose Wind Tunnels

Although most of the tunnels are built for aviation industry, some tunnels have been used for other areas with minor modifications. Below are the areas how the wind tunnels are used for different applications.

(a) People

The wind tunnels are used for increasing the performance of bikers or skiers. The athlete can learn which posture gives him the minimum drag. In addition, ski jumpers can learn to have best lift to drag ratio to glide further.

(b) Wind Power Devices

The wind tunnels are also used to simulate the conditions that a wind power device can have during its life cycle. Experiments in the wind tunnel have significant importance for the integrity of the system. Also, the efficiency is worth studying to increase the energy taken for the same size of wind mills.

(c) Solar Collectors, Radar Antenna and Satellite Dish

The tests are performed to obtain the loads on the solar collectors. The shapes of the collectors have different aerodynamic pressure distribution in different wind conditions that must be considered during design of the foundation. In addition, elastic movements under the influence of the air flowing around the collector may change the focal point which will decrease the efficiency. Radar antennas and satellite dishes are also subject to wind tunnel testing for obtaining the data for structural integrity.

(d) Bridges

The natural winds flowing over the bridges have great importance. The wind tunnels supply useful information for long span bridges. Even at very 12 small wind speeds,the aerodynamic excitation can lead fatigue crack grew on the beams of the bridge structure. The effects are caused by a phenomenon called vortex shedding. The result of the vortex shedding may be seen as vertical movement or torsional oscillations that can be a very serious problem for the bridges. Also, flutter instability is another problem which must be investigated before the bridge has been built in wind tunnels.

1.3. History and Background

Wind tunnel has a long history. Many scientist studies about the wind tunnel for its development. At first a British military engineer and mathematician Benjamin Robins (1707–1751) invented a whirling arm apparatus to determine drag and did some of the first experiments in aviation theory.

After some years later a scientist Sir George Cayley (1773–1857) also used a whirling arm to measure the drag and lift of various airfoils. His whirling arm was 5 feet (1.5 m) long and attained top speeds between 10 and 20 feet per second (3 to 6 m/s).
Then Otto Lilienthal used a rotating arm to accurately measure wing airfoils with varying angles of attack, establishing their lift-to-drag ratio polar diagrams, but was lacking the notions of induced drag and Reynolds numbers.

However, the whirling arm does not produce a reliable flow of air impacting the test shape at a normal incidence. Centrifugal forces and the fact that the object is moving in its own wake mean that detailed examination of the airflow is difficult. Francis Herbert Wenham (1824–1908), a Council Member of the Aeronautical Society of Great Britain, addressed these issues by inventing, designing and operating the first enclosed wind tunnel in 1871. Once this breakthrough had been achieved, detailed technical data was rapidly extracted by the use of this tool. Wenham and his colleague John Browning are credited with many fundamental discoveries, including the measurement of l/d ratios, and the revelation of the beneficial effects of a high aspect ratio. Konstantin Tsiolkovsky built an open section wind tunnel with a centrifugal blower in 1897, and determined the drag coefficients of flat plates, cylinders and spheres.

Danish inventor Paul la Cour applied wind tunnels in his process of developing and refining the technology of wind turbines in the early 1890s. Carl Rickard Nyberg used a wind tunnel when designing his Flagon from 1897 and onwards.

In a classic set of experiments, the Englishman Osborne Reynolds (1842– 1912) of the University of Manchester demonstrated that the airflow pattern over a scale model would be the same for the full-scale vehicle if a certain flow parameter were the same in both cases. This factor, now known as the Reynolds number, is a basic parameter in the description of all fluid-flow situations, including the shapes of flow patterns, the ease of heat transfer, and the onset of turbulence. This comprises the central scientific justification for the use of models in wind tunnels to simulate real-life phenomena. However, there are limitations on conditions in which dynamic similarity is based upon the Reynolds number alone.
The Wright brothers' use of a simple wind tunnel in 1901 to study the effects of airflow over various shapes while developing their Wright Flyer was in some way’s revolutionary. It can

be seen from the above, however, that they were simply using the accepted technology of the day, though this was not yet a common technology in America.
In France, Gustavo Eiffel (1832–1923) built his first open-return wind tunnel in 1909, powered by a 50 -kW electric motor, at Champs-de-Mars, near the foot of the tower that bears his name. Between 1909 and 1912 Eiffel ran about 4,000 tests in his wind tunnel, and his systematic experimentation set new standards for aeronautical research. In 1912 Eiffel's laboratory was moved to Auteuil, a suburb of Paris, where his wind tunnel with a two- meter test section is still operational today. Eiffel significantly improved the efficiency of the open-return wind tunnel by enclosing the test section in a chamber, designing a flared inlet with a honeycomb flow straightener and adding a diffuser between the test section and the fan located at the downstream end of the diffuser; this was an arrangement followed by a number of wind tunnels later built; in fact the open-return low-speed wind tunnel is often called the Eiffel type wind tunnel.

1.3.1. Literature review

Wind tunnels have long served as fundamental tools for aerodynamic research and development. Low subsonic wind tunnels, which operate at Mach numbers below 0.3, are primarily used for testing small-scale models to study flow characteristics, pressure distributions, and drag forces at relatively low airspeeds. This review examines existing literature relevant to the design, calibration, and performance evaluation of low subsonic wind tunnels.

Velocity calibration is a key component of wind tunnel performance tests. Standard calibration techniques involve using a pitot-static probe and comparing results with theoretical or numerical expectations. Mohamed and Hassan (2007) demonstrated that precise calibration ensures the reliability of aerodynamic coefficients obtained during testing.

Modern wind tunnel experiments often incorporate advanced instrumentation such as multi-hole pressure probes, hot-wire anemometers, and PIV (Particle Image Velocimetry). Each method offers unique advantages in characterizing flow fields. Work by Ligrani and Bradshaw (1987) showed how turbulence measurements can be affected by probe positioning and flow conditions. low-speed wind tunnel for drone testing, highlighting challenges in achieving sufficient Reynolds numbers and flow quality in small setups. The performance of a low subsonic wind tunnel is largely determined by its ability to maintain a uniform, low-turbulence flow. Key areas of interest in the literature include flow quality control, velocity calibration, and instrumentation. Continued developments in measurement technology and computational methods are enhancing the accuracy and usefulness of wind tunnel performance tests.

1.4 Objectives

1. To Repair the Subsonic Wind Tunnel Present at SU Lab.
2. To test the Performance of the Subsonic Wind Tunnel.

CHAPTER II

WIND TUNNEL COMPONENTS

2.1. Wind Tunnel Components

Wind tunnel has different types of components such as contraction cone, settling chamber or flow straightener, diffuser and fan section etc. Those components are shortly described here.

2.1.1. Contraction Cone

The contraction accelerates the flow to the test section, further reducing any variations in velocity. A contraction design satisfying all criteria will be such that the separation is just avoided and the exit non-uniformity is equal to or less than the maximum tolerable level for the desired application in shortest possible length. The contraction starts with the selection of contraction ratio 6. For smaller tunnels the contraction ratio should be in between 6 to 9. The length of the contraction should be in between 0.15 R to R. R being the hydraulic radius of the contraction section inlet.

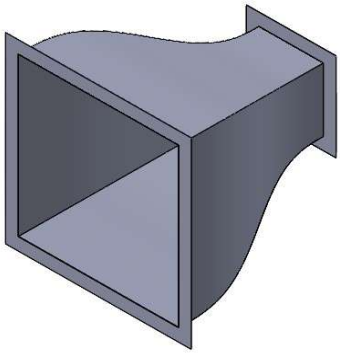


Figure 2.1: Contraction Cone

2.1.2. Diffuser

Diffuser is the component of the wind tunnel which reduces the speed by increasing the cross-sectional area. It can be circular, rectangular or polygonal. When it is used just after the test section, the geometry of the cross section may change throughout the diffuser.
When diffusers are placed just after the high-speed sections, the design must be carefully examined. Any flow separation in the diffuser will affect other components and performance of the wind tunnel. In diffuser design, a parameter called equivalent cone angle is used.
Equivalent cone angle is calculated by using a hypothetical cone, having the same area for inlet and exit of the diffuser. A representative drawing is given in Figure

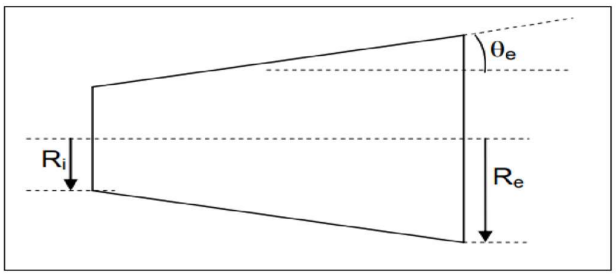


Figure 2.2: Diffuser Geometry and Notation

Equivalent cone angle must be less than some value for the diffuser to operate as designed. When the diffuser has sharp corners, the equivalent cone angle will have 0.5°~1.0° lower limit than for the rounded cross sections. The equivalent cone angle must be less than 4-5 for diffusers

2.1.3. Test Section

The wind tunnel design starts with deciding the test section keeping an eye on the accessibility and installation of the test model and instrumentation. From the experimental investigations, it has been found that the blockage has an almost negligible effect on test results when it is about 10%^2. The test chamber length has to be in the range of 0.5-3 times its hydraulic diameter. The test chamber is being designed to test the scaled model of a silo with length 925 mm and minimum L/D ratio 2. In order to have a blockage ratio less than 10% a square test chamber of 925 mm side is chosen. The test will be carried out at a flow speed of 40 m/s. The pressure loss coefficient goes on increasing with increase in the test section Figure 8. So, the length of test section should be as small as possible. The length of the test chamber was set to 1.0 m i.e. 2.0 times the hydraulic diameter of the test section.

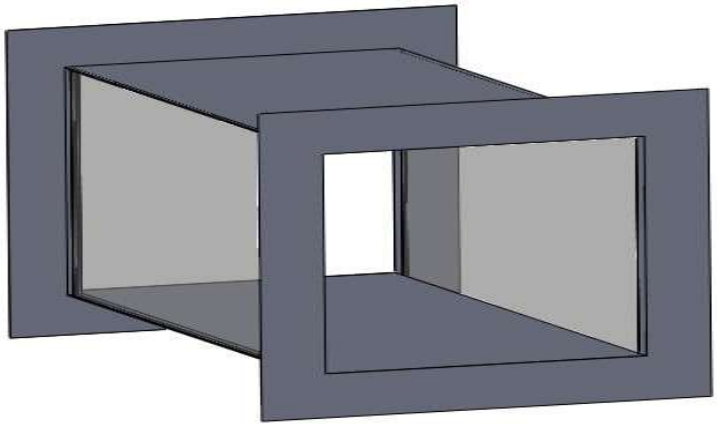


Figure 2.3: Test Section

2.1.4. Settling Chamber

The aim of a settling chamber containing honeycombs and screens is to reduce the flow turbulence before it enters the cone. A selection of honeycomb and screens for a wind tunnel is very much dependent on the test type to which the tunnel is intended. The settling chamber cross-sectional area matches the dimensions of contraction cone.

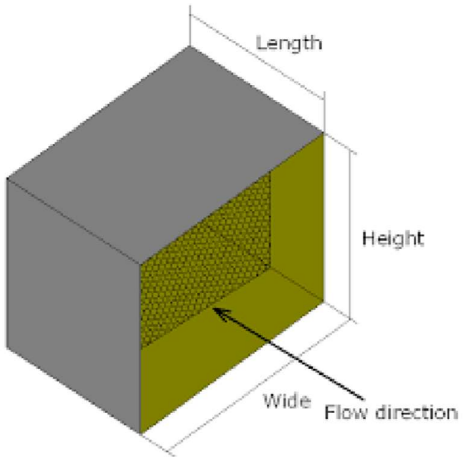


Figure 2.4: Settling Chamber

2.1.5. Honeycomb

The honeycomb removes the swirl from the incoming flow and minimizes the variation in both mean and fluctuating velocity. It should have sufficient flexural rigidity to withstand applied forces during operation without significant deformation. The primary design parameter for honeycomb is the ratio of length to cell hydraulic diameter (L/D) and porosity (flow area upon total area). Recommended L/D ratio for honeycomb is 6 to 8 with porosity nearly 0.8. A circular shaped honeycomb,2 mm thick and 18 mm dia. Figure 2.5 shows an arrangement of Honeycomb and screens in settling chamber**.**

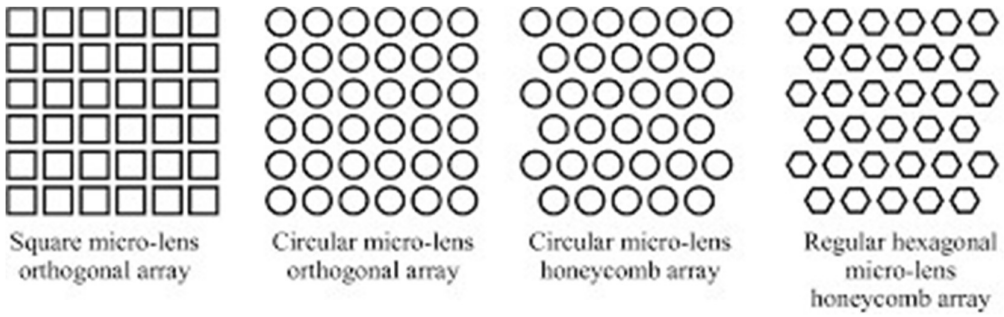


Figure 2.5: Honeycomb

2.1.6. Screen

Tensioned screens are placed in the settling chamber for the reduction of turbulence level of incoming flow, this breaks up the large-scale turbulent eddies into a number of small-scale eddies that subsequently decay. To be effective in reducing turbulence a screen must have porosity in the range 0.58–0.89. Screen porosity values over 0.8 are not suitable for good turbulence control while values below 0.58 lead to flow instability10. A clearance of 0.2 times the settling chamber diameter is required between the screens. Experimentally it is found that bar turbulence control can be achieved when the finest screen is placed farthest downstream.



Figure 2.6: Screen and Mesh

2.1.7 Turbo Machinery

Turbo Machinery, in mechanical engineering describes machines that transfer energy between a rotor and a fluid including both turbines and compressor. While turbine transfers energy from a fluid to rotor and compressor transfer energy from a rotor to fluid. The American Society of Mechanical Engineers (ASME) uses specific ratio that is the ratio of discharge pressure to suction pressure to classify the air moving equipment.

2.1.8 Axial Flow Fan Types

The types of axial fans can be classified in three groups. They are propeller fans, tube axial fans, vane axial fans. Another application is multi stage axial fans. By use of multistage axial fans, it is possible to have higher pressure rise with the same volumetric flow rate.

2.1.9. Propeller Fans

Propeller fans are used for high flow rates with little pressure rise. Generally, they are not combined with extensive ducts or systems that require high pressure increase. They are considered to be inexpensive because of relatively simple construction. Their maximum efficiency is achieved near free delivery. They are used for rooftop ventilation or as panels mounted on walls of structures. They can generate flow in reverse direction, which is suitable for ventilation applications.

The energy efficiency is low compared with other types of axial flow fans. In addition, they are considered to be noisy. The drive system can be either direct drive that is the motor is directly coupled to impellers or indirect drive that is the mechanical motion is transferred by belt to the hub.

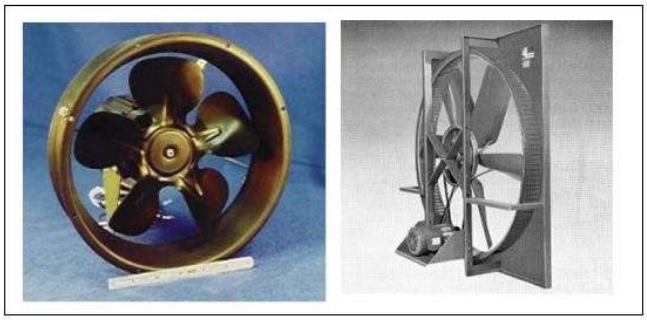


Figure 2.7: Direct Drive and Belt Driven Propeller Fan

2.1.10 Tube Axial Fans

Tube axial fans are essentially propeller fans placed inside a cylindrical shell, which is about one diameter long generally. Tube axial fans can generate higher pressure rise and operate at better efficiencies than the propeller fans. They are used in ducted HVAC applications. It can generate flow in reverse direction, which is very useful for most ventilation 18 applications. The hub diameter is about 30% to 50% of the impeller diameter. Tube axial fan performs better when used as exhausting fans.

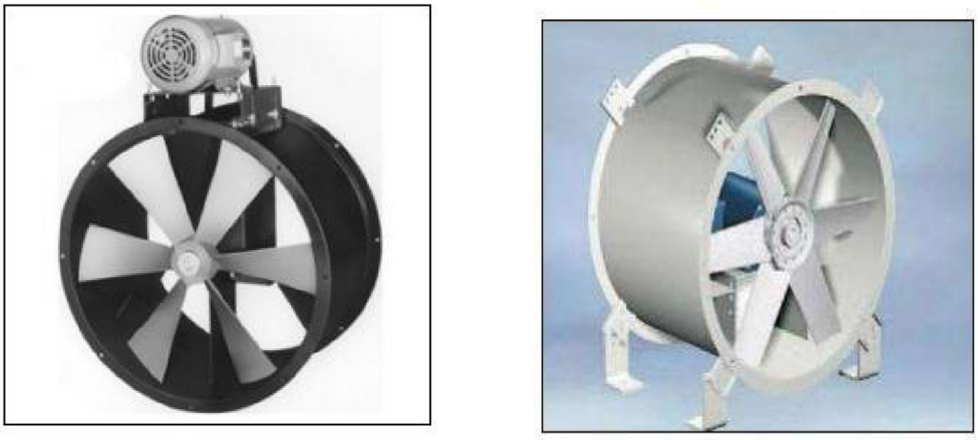


Figure 2.8: Direct Drive and Belt Drive Tube Axial Fan

2.1.11 Vane Axial Fans

A vane axial fan is very similar to a tube axial fan, except it has additional guide vanes to direct the flow into a more suitable path to the impellers or to remove the swirl component of the velocity to have additional gain of static pressure. The hub diameter is 50% to 80% percent of the impeller diameter. Vane axial fans can generate high volumetric flow rates and fairly high static pressure rise compared to the previous axial fan types. The shell is generally one diameter long. In some application, the motor is buried in a tail cone that increases aerodynamic efficiency by a controlled diffusion of the flow. The tolerance between the tip of the impeller blades and the shell has an effect on efficiency. Also using airfoil blades has a positive effect on performance and efficiency. The overall efficiency of well-designed vane axial fan can exceed 85.



Figure 2.9: Direct Drive Vane Axial Fan

Here direct drive vane axial fan will be used. Because it has higher efficiency than other types of fans. Direct drive will be used because belt drive has some losses for the friction of belt. Reducing the losses and to get highest efficiency direct drive vane axial fan is suitable for the drive system.

2.2Load Transmission

As the wind tunnel test section can horizontal and the wind direction can perpendicular to the direction of Earth's gravity, the wind axes are parallel to ground axes. This means that the lift can parallel direction to Earth's gravity and drag can perpendicular to it. This is the reason the lift measurement will be done vertically and drag measurement will be done horizontally. Wind tunnel test section with wind axis and forces. Before doing the measurements the horizontal and vertical load must be separated and transmitted to the sensors minimizing interferences between them in order to read the most near to the real value.

2.3 The Balance Bar

The balance bar is balancing the test object and help to get accurate data from test object. It's a threaded rod, which is located by nut & washer and for moving object we are used wooden handle.

The balance bar for the test object is 500 mm long, 10 mm diameter MS thread rod. It can be attached with the test object. It would be attached with balanced support plates. It can hold the test object. The balance bar attached with the handle by changing the angle of attack.



Figure 2.10: Balance Bar.

2.4 Test object

A wind tunnel testing machine consists of a closed tubular passage with the object under test mounted in the middle and powerful fan system moves air past to the object. The wing (airfoil) Provides lift by creating a situation where the pressure above the wing is lower than the pressure below the wing.

NACA 0024 airfoil would be used in the test section. Airfoil length can be 120 mm long and width can be 150 mm. Airfoil can be a shape of a body that is mainly placed in a stream of air flow or water flow in order to produce useful aerodynamic effect depending on the usage and need.
Test object would be attached with the force balance. Test object would be attached in the center point of the test section. Force balance would be holding the airfoil in proper place.

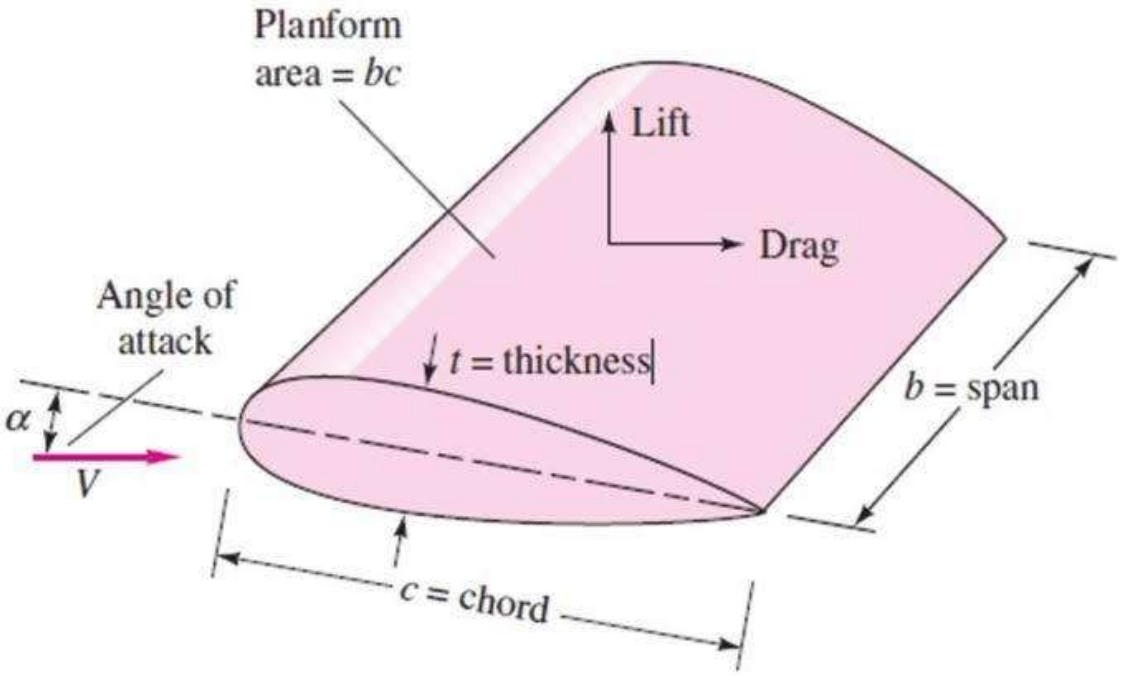


Figure 2.11: Schematic Diagram of Test object

Airfoil drag can be determined by integrating the total pressure deficit in the wake created by a wing model. In some wind tunnel tests, the model is instrumented to provide diagnostic information about the flow of air around the model.

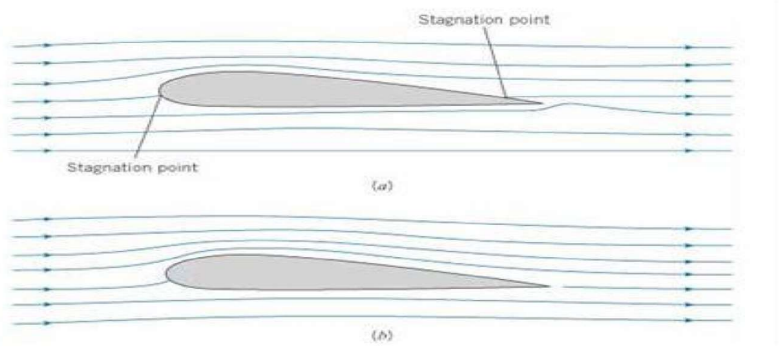


Figure 2.12: Patterns of Flow Around Airfoil

A wind tunnel consists of a closed tubular passage with the object under test mounted in the middle and powerful fan system moves air past to the object. The airplane generates lift using its wings, the cross-sectional shape of the wing is called an airfoil

2.5 Sensor

To have the digital data from the experiments different types of sensors are needed to set in the test section which have to be connected with the test object. In this section different types of sensors are described which is essential to have a full experimental setup.

2.5.1 Digital Anemometer AS836 (Smart Sensor)

Digital Anemometer is a device which is used for measuring wind speed & temperature of air. These device has two part one is anemometer and another is a small fan which is connected with cable. This fan is located inside the test section and digital anemometer located outside the test section.



Figure 2.13: Digital Anemometer

CHAPTER III

AIRFOIL MODEL, DRAG FORCE & LIFT FORCE

3.1. Airfoil Model

Basically, wind tunnel is used for test an object. This object should have some losses. Here an airfoil is used for loss calculation. As an airfoil NACA 0024 is used.

Airfoil: NACA 0024
NACA 0024 is a NACA 4 -digit airfoil. NACA means National Advisory Committee for Aeronautics. The first digit expresses the camber in the percentage of chord, the second digit gives the location of maximum camber point in tenths of a chord and the last two digits gives the thickness in the percentage of airfoil or object.

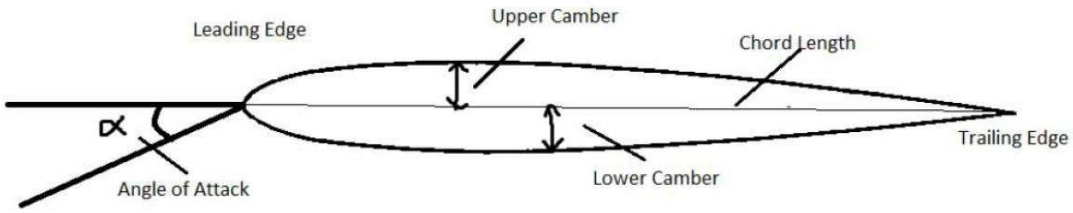


Figure 3.1: Basic Nomenclature of an Airfoil

Model calculation:

Let, NACA 4 digit airfoil model is NACA MPXX

NACA= National Advisory Committee for Aeronautics

M = % of the chord length is the maximum camber

P = % of the chord is the location of maximum camber

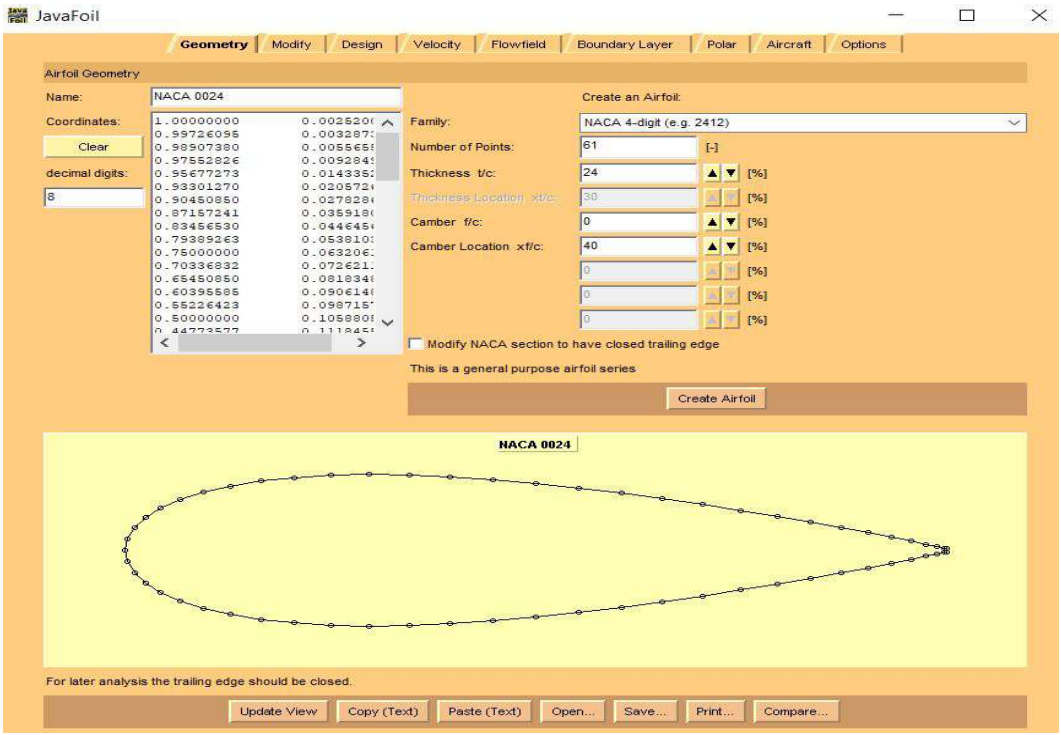
XX = % of the chord length is the maximum thickness of the airfoil

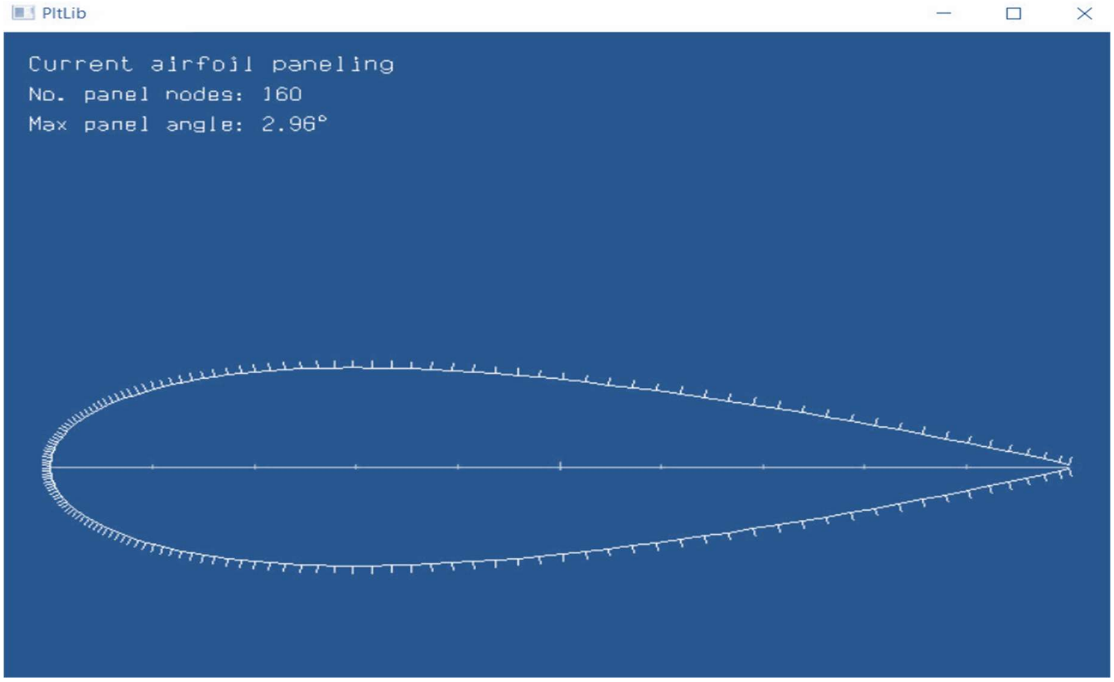
As chord line and camber line are same so maximum camber is zero, for this reason, M=0 and P=0.

As chord length is 150mm and airfoil thickness is 36mm

So, XX= (36/150)*100
= 24%

Our using airfoil model is NACA0024 like as below by using javafoil software-





3.2 Drag Force

The resistance offered by a fluid, like air and water, is called the drag force or drag when an object moves through it. It acts in a direction opposite to the object's motion. It is generated by the relative velocity between the solid object and the fluid. Whether the object or fluid is moving, drag occurs as long as there is a difference in their velocities. Because it is resistant to motion, drag tends to slow down the object. An effective way to reduce it is to alter the shape of the object and make it streamline.

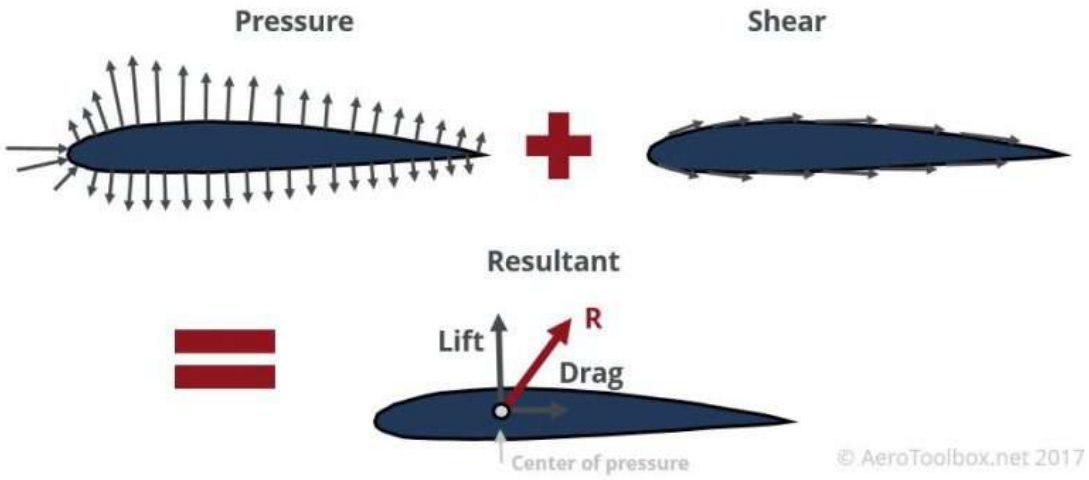


Figure 3.2: Relation between Drag and Lift Force

<u>Drag force</u> , $D_f = ½ \rho V^2 S C_D$ ……..(1)
Where,

V= Up stream air velocity

S= Wing Area/ Object Area

ρ = Air density=1.1839 Kg/m^3

C_D= Drag force Coefficient

3.3 Drag coefficient:

The **drag force coefficient** (often denoted as Cd) is a **dimensionless number** that quantifies the drag or resistance of an object in a fluid environment, such as air or water. It is defined as the ratio of the drag force to the dynamic pressure times the reference area of the object. The formula for calculating drag force is given by:

Coefficient of drag, $C_D = \frac{2D}{\rho \times V\text{^}2 \times S}$ ……(3)

3.4 Lift Force

When a gas flows over an object, or when an object moves through a gas, the molecules of the gas are free to move about the object; they are not closely bound to one another as in a solid. Because the molecules move, there is a velocity associated with the gas. Within the gas, the velocity can have very different values at different places near the object. Bernoulli's equation, which was named for Daniel Bernoulli, relates the pressure in a gas to the local velocity; so as the velocity changes around the object, the pressure changes as well. Adding up (integrating) the pressure variation times the area around the entire body determines the aerodynamic force on the body. The lift is the component of the aerodynamic force which is perpendicular to the original flow direction of the gas. The drag is the component of the aerodynamic force which is parallel to the original flow direction of the gas. Now adding up the velocity variation around the object instead of the pressure variation also determines the aerodynamic force. The integrated velocity variation around the object produces a net turning of the gas flow. From Newton's third law of motion, a turning action of the flow will result in a re-action (aerodynamic force) on the object. *So **both** "Bernoulli" and "Newton" are correct*. Integrating the effects of either the pressure or the velocity determines the aerodynamic force on an object. We can use equations developed by each of them to determine the magnitude and direction of the aerodynamic force.

Lift force $L = \frac{1}{2} C_L \rho V^2 S$ …….(6)

Where,

C_L= Lift Coefficient

3.5 Lift Coefficient,

The **lift coefficient** is a number that aerodynamicists use to model all of the complex dependencies of shape, inclination, and some flow conditions on lift. This equation is simply a rearrangement of the lift equation where we solve for the lift coefficient in terms of the other variables. The lift coefficient C_l is equal to the lift **L** divided by the quantity: density **rho (ρ)** times half the velocity **V** squared times the wing area **S**.

$$CL = L/(0.5\times\rho\times V\text{^}2\times S) \quad \text{…….(7)}$$

The quantity one half the density times the velocity squared is called the dynamic pressure **q**.

So $C_L = \frac{L}{qS}$

The lift coefficient then expresses the ratio of the lift force to the force produced by the dynamic pressure times the area.

Here is a way to determine a value for the lift coefficient. In a controlled environment (wind tunnel) we can set the velocity, density, and area and measure the lift produced. Through division, we arrive at a value for the lift coefficient. We can then predict the lift that will be produced under a different set of velocity, density (altitude), and area conditions using the lift equation.

The lift coefficient contains the complex dependencies of object shape on lift. For three dimensional wings, the downwash generated near the wing tips reduces the overall lift coefficient of the wing. The lift coefficient also contains the effects of air viscosity and compressibility. To correctly use the lift coefficient, we must be sure that the viscosity and compressibility effects are the same between our measured case and the predicted case. Otherwise, the prediction will be inaccurate.



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

Cl contains all the complex dependencies and is usually determined experimentally.

CHAPTER IV

PERFORMANCE TEST & CALCULATION

4.1. Flow parameter Calculation

Flow parameter should be calculated to determine the external source by which required velocity and pressure are obtained.

4.1.1. Reynolds Number

The Reynolds number is calculated for an airfoil or paratrooper length of 250 mm.
Reynolds number= Inertia force / Viscous force.

$$Re = \rho v L / \mu \quad (4.1)$$

Where,

Re = Reynolds number
ρ = Density of air in kg/m^3
L = Chord length of the airfoil in meter
V = Velocity of the test section in m/s
μ = Dynamic viscosity of air in Pa.s

Now at room temperature (25° C)

$\rho = 1.1839$ Kg/m^3
$\mu = 1.86*10^{-5}$ Pa. s

Here,

L= 0.12 m
V= 7.02 m/s

From equation (4.1)

$$Re = \frac{1.183 \times 7.02 \times 0.12}{1.86 \times 10^{-5}}$$

$$= 53578$$

4.1.2. Mach Number

Mach number is calculated for a fixed velocity 7.02 m/s in the test section.
Mach number = Object speed / Speed of sound

$$M = V / C \quad (4.2)$$

Where,

M = Mach number
V = Velocity of flow in m/s
C = Velocity of sound in m/s

Here at 25° C

C = 346 m/s
And V = 7.02 m/s

Now from equation (4.2)

$M = 7.02/ 346$

$= 0.02$

$M < 1$,So flow is subsonic

4.1 Table: Different Flow Region Based on Mach Number:

Types of flow	Mach number
Subsonic	M < 1
Sonic	M = 1
Supersonic	1 < M < 3
Hypersonic	M > 4

4.2 Drag Force and Lift Force Calculations:

Table 4.2: Data generated by javaFoil for NACA 0024 model

Angle of attack α	Coefficient of Lift force C_L	Coefficient of Drag force C_d	Pitching moment Coefficient Cm 0.25	Pressure coefficient Cp*	Moment coefficient about reference Point Mcr
0	0	0.0277	0	-0.846	0.634
10	1.151	0.05281	-0.029	-3.275	0.404
20	1.769	0.1989	-0.041	-8.628	0.267
30	1.79	0.46273	-0.047	-16.02	0.199
40	1.41	0.85141	-0.049	-24.229	0.164
50	1.067	1.33658	-0.04	-32.264	0.143
60	0.844	1.88676	-0.034	-39.156	0.13
70	0.7	2.34328	-0.025	-47.107	0.118
80	0.619	2.65725	-0.013	-53.047	0.112
90	0.672	2.08515	0	-55.912	0.109

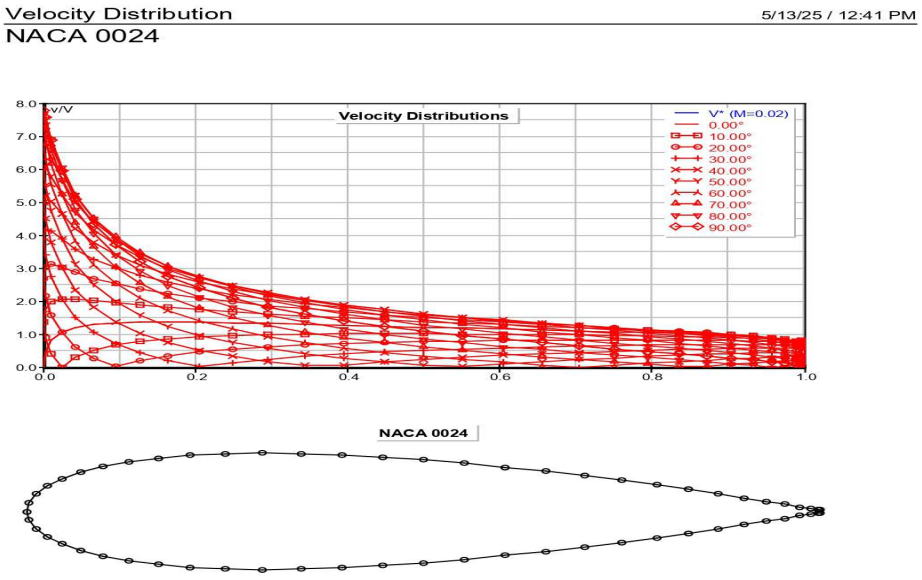


Figure 4.1: Velocity Distributions

Sample calculation:

Drag force

From equation no we know that,

Drag force, $D_f = \frac{1}{2} \rho V^2 S C_D$

$D_f = \frac{1}{2} \times 11.61 \times 6.87^2 \times 0.018 \times 0.053$

$= 0.26$ N

Lift Force

From equation no (6) we get,

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

$= 0.5 \times 1.15 \times 11.61 \times 6.87^2 \times 0.018$

$= 5.67$ N

We calculate drag force for angle of attack 10 degree.

Where

Air density $\rho = 1.1839$ Kg/m^3

$= 11.61$ N/m^3

Upstream velocity V = 7.02 m/s

Wing area S = chord length× wing span

$=0.12 \times 0.15$

$=0.018$ m^2

Note: Drag force coefficient C_D and Lift force Coefficient C_L value has taken from Table no 4.2 at angle of attack 10 degree.

Table 4.3: Drag force and lift force with coefficient

Angle of Attack (°)	Coefficient of Lift force CL	Coefficient of Drag force Cd	Upstream Velocity (m/s)	Downstream Velocity (m/s)	Wing area(S) m^2	Drag force (N)	Lift force (N)
0	0	0.03	7.02	4.5	0.018	0.14	0.00
10	1.15	0.05	6.87	5.1	0.018	0.26	5.68
20	1.77	0.20	6.79	5	0.018	0.96	8.52
30	1.79	0.46	6.66	4.7	0.018	2.14	8.30
40	1.41	0.85	6.58	1	0.018	3.85	6.38
50	1.07	1.34	6.4	1.5	0.018	5.72	4.57
60	0.84	1.89	6.49	2.5	0.018	8.30	3.71
70	0.70	2.34	6.5	2	0.018	10.34	3.09
80	0.62	2.66	6.43	1.8	0.018	11.48	2.67
90	0.67	2.09	6.48	0.3	0.018	9.15	2.95

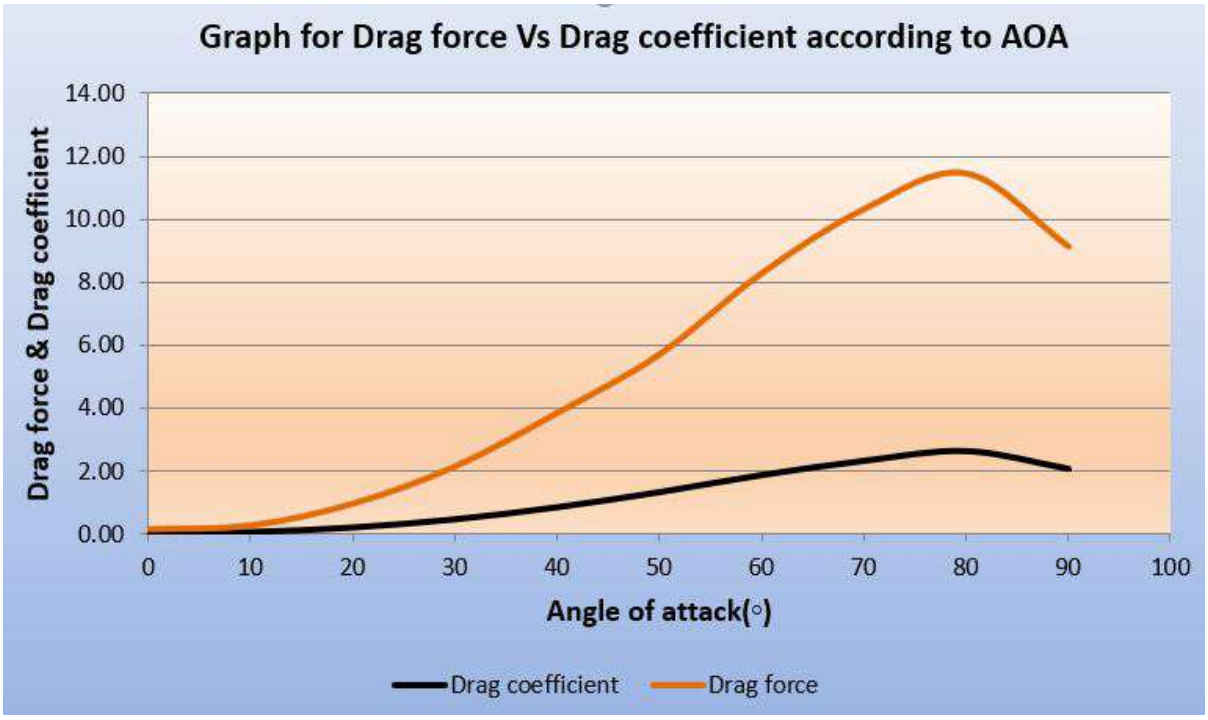


Figure 4.2: Drag force VS Drag Coefficient.

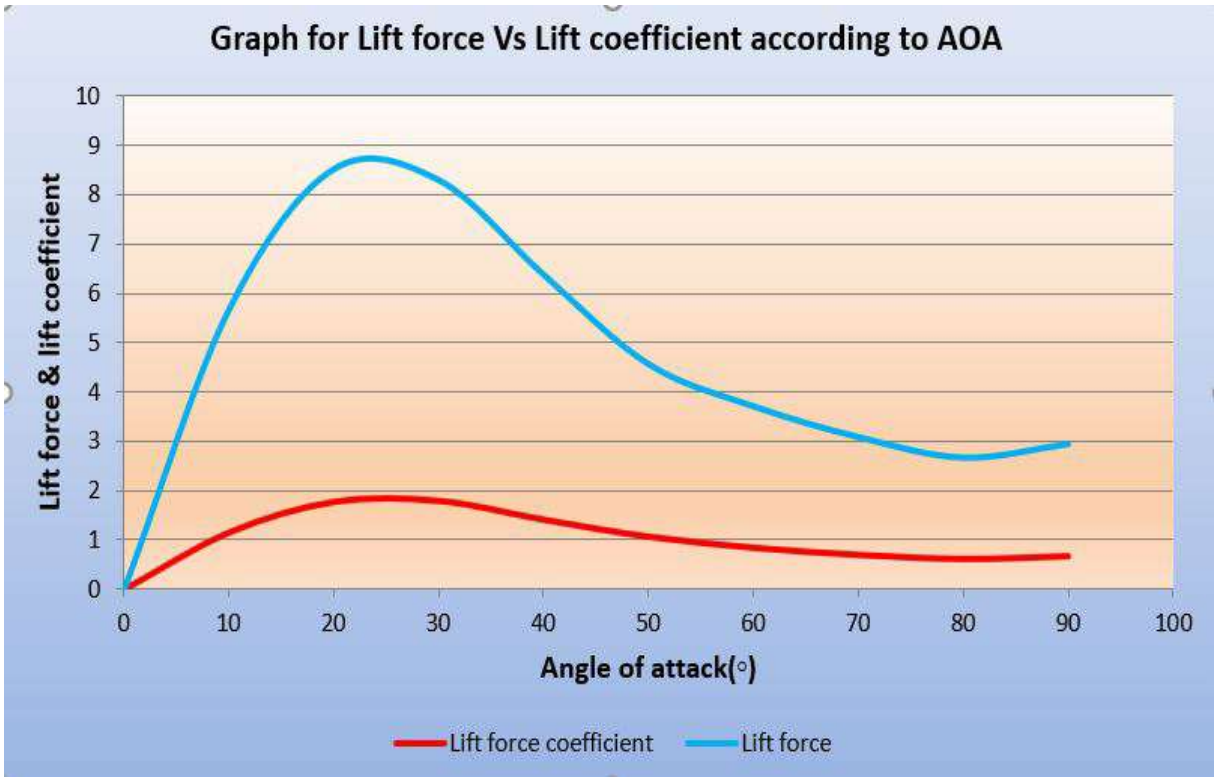


Figure 4.3: Lift force VS Lift Coefficient

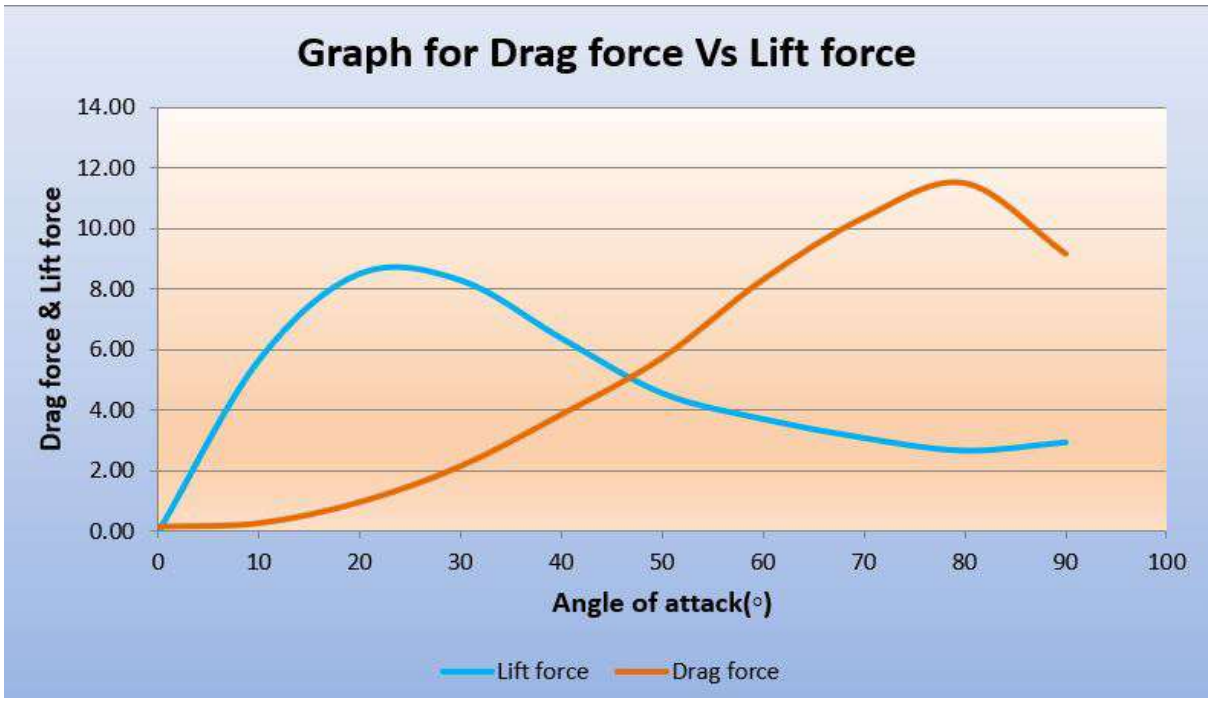


Figure 4.4: Drag force VS Lift force

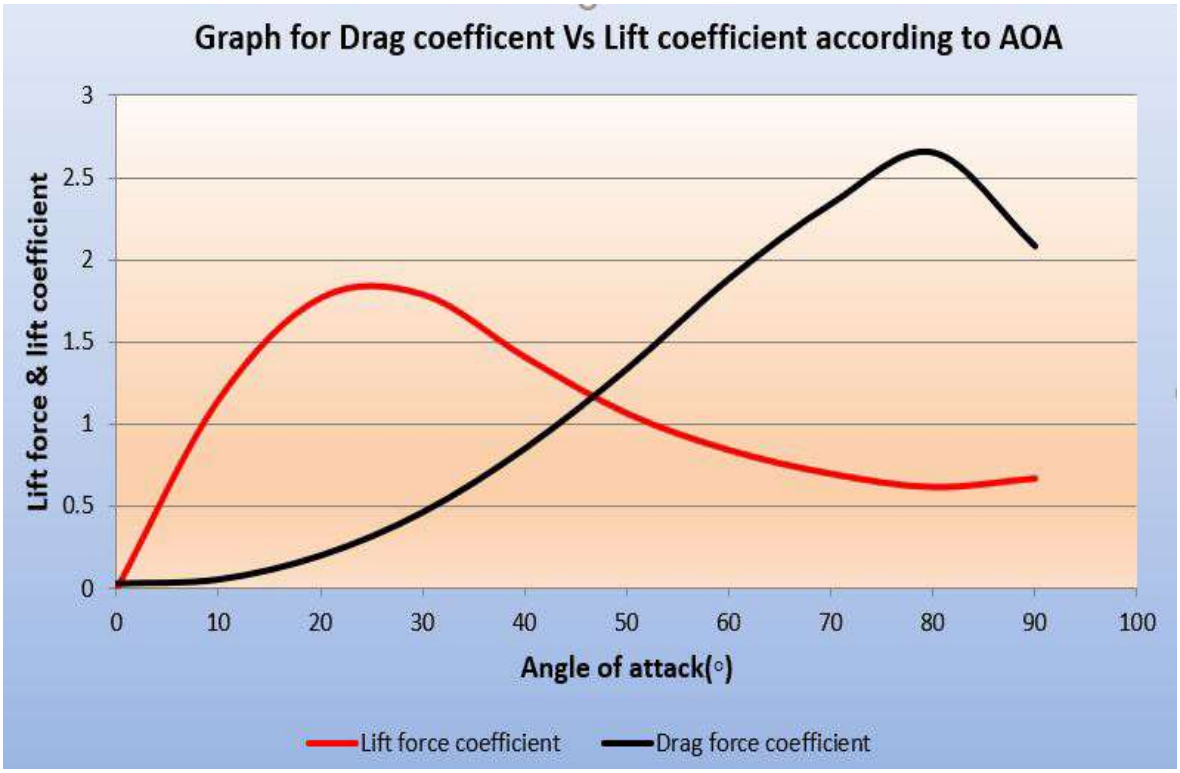
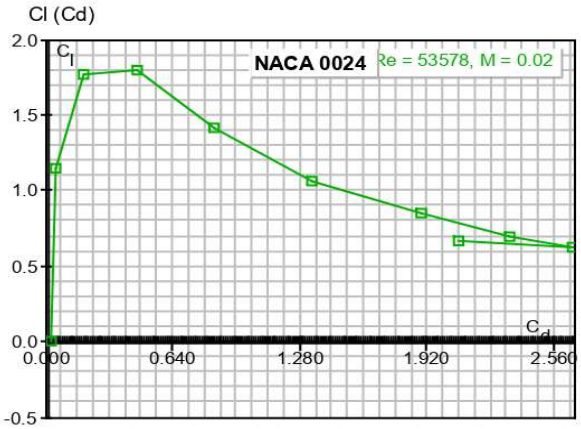
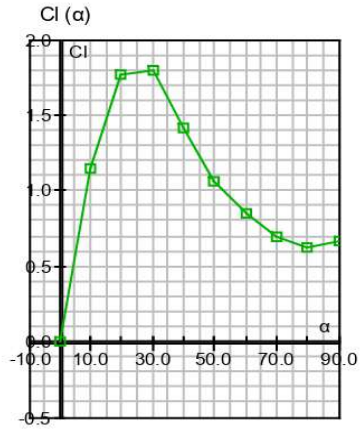
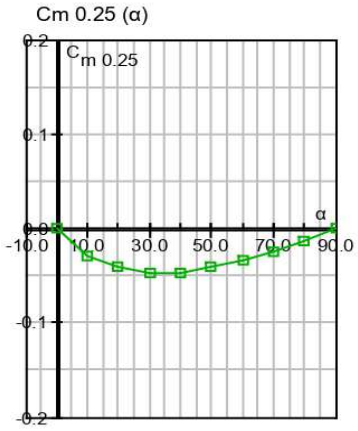


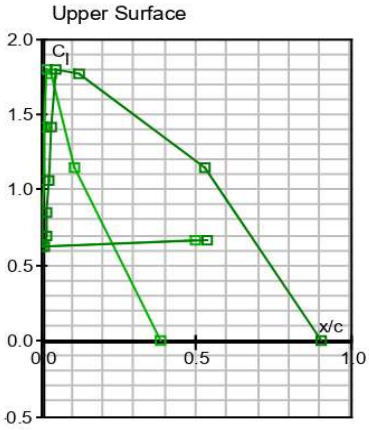
Figure 4.5: Drag force Coefficient VS Lift force Coefficient

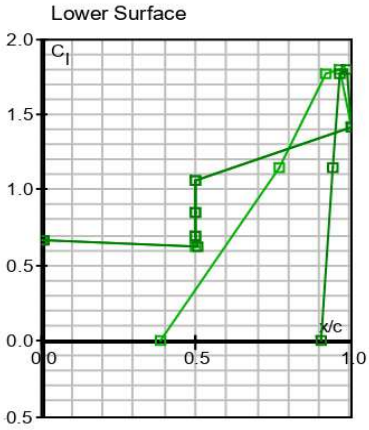
NACA 0024











Created by JavaFoil © 2001-2017 © Martin Hepperle

4.3. Result Analysis

Actual lift and drag forces along with their corresponding co-efficient:

The lift forces follows a trend similar to the lift coefficient peaking near 25 degree AOA before declining due to stall. The drag forces remains relatively low at small AOAs but increases significantly beyond 50 degree, becoming dominant at high AOAs (70-80) degree.

The intersection point of the lift and drag force lines occurs beyond 60 degree, beyond which drag overtakes lift, indicating aerodynamic condition for most airfoil applications.

The drag coefficient (Cd) steadily rises with AOA, while the lift coefficient (Cl) shows rise and fall pattern, again reflecting stall behavior. The optimal AOA range for maximum aerodynamic efficiency is around 20 -25 degree, where the lift is maximized ,and drag is still relatively low.

CHAPTER V

CONCLUSIONS

5.1. Conclusion

The performance test of the low subsonic wind tunnel successfully validated its operational capabilities and measurement accuracy within the expected design parameters. The test confirmed uniform flow distribution in the test section, acceptable turbulence intensity levels, and consistent velocity profiles. Calibration of instrumentation showed that velocity measurements were within acceptable error margins, and the flow quality was suitable for aerodynamic testing of small-scale models. These results indicate that the wind tunnel is a reliable tool for academic research and student experiments involving low-speed aerodynamics. Key objectives including the establishment of uniform flow, pressure distribution analysis, and visualization of flow patterns were effectively achieved. The experimental results were consistent with theoretical predictions and CFD simulations, validating both the tunnel and design and its operational integrity. Additionally, the project enhanced understanding of aerodynamic forces such as lift and drag, and the impact of model geometry on airflow characteristics. Overall, the project provided a hands-on learning experience in wind tunnel design, flow measurement techniques, and data interpretation, forming a strong foundation for further research in aerodynamics and fluid dynamics.

REFERENCES

[1] "Wind Tunnel - History," [https://en.wikipedia.org/wiki/Wind_tunnel]

[2] M.K.P.a.A. K Samanta “Design of Low Cost Open Circuit Wind Tunnel – A Case Study,’’ India Journal Of Science and Technology, August 2016

[3] F. Çevik, "Design of an Axial Flow Fan for a Vertical Wind Tunnel for Paratroopers, " A Thesis Submitted to the Graduate School of Natural and Applied Science of Middle East Technical University

[4] Airfoil Model [https://youtu.be/9YBacXvw7Qo?si=pezHfdO0P30knrpG]

[5] Lift ,Drag & Moment Coefficient[https://aerotoolbox.com/lift-drag-moment-coefficient]

[6] Reynolds Number [Study.com]

[7] Mach number [https://www.grc.nasa.gov]

[8] https://www1.grc.nasa.gov/

[9] JavaFoil software

[10] Fundamental of aerodynamics sixth edition by Jon Anderson.

[11] C. B. a. J. M. Louis Cattafesta,"Fundamentals of Wind-Tunnel Design,” Department of Mechanical and Aerospace Engineering, University of Florida, Gainesville, FL,USA.

[12] Y. J. H. B. a. R. D. Mehta, "Contraction Design for Small Low-Speed Wind Tunnels," p. 1 & 2, April 1988

[13] R. L. B. a. J. T. B. Langley, "A Generalized Theory for the Design of Contraction Cones and other Low-Speeds Ducts," Research Center Hampton, vol. 23365, p. 1 & 3, November 1972