EXPERIMENTAL DESIGN AND TESTING OF WIND TUNNEL

[1]K.SAI ADITHYA [2]M.MANI KANTA [3] CH.RAGHU RAM [4]Y.YOGENDHRA [5]B.USHA RANI

[1] B.Tech Student , Department of Mechanical Engineering , NSRIT,Visakhapatnam,India, [2] B.tech Student , Department of Mechanical Engineering , NSRIT,Visakhapatnam,India ,[3] B.tech Student , Department of Mechanical Engineering , NSRIT,Visakhapatnam,India ,[4] B.tech Student , Department of Mechanical Engineering, NSRIT,Visakhapatnam,India , [5]Asst.Professor, Department of Mechanical Engineering, NSRIT,Visakhapatnam,India.

ABSTRACT

A Wind Tunnel is a tool used in aerodynamic research to study the effects of air moving past solid objects. The project focuses on aerodynamics design of a Wind Tunnel to simulate subsonic flows. Our specific aims were to design and construct a user-friendly Wind Tunnel facility, to adapt Wind Tunnel experiments such as flow visualization and measuring lift, drag to DBT (Design, Build and Test) projects, to assess and disseminate results. The proposed tunnel was designed based on flow theories to obtain a preliminary design of subsonic tunnel. The tunnel is open type Wind Tunnel. The design consists of CATIA modelling of the nozzle, Test Section and diffuser and CFD analysis is carried out using ANSYS Fluent.

KEYWORDS: Wind tunnel, Subsonic flow, CATIA,CFD Analysis,Diffuser

1. INTRODUCTION

A Wind Tunnel is a tool used in aerodynamic research to study the effects of air moving past solid objects. A Wind Tunnel consists of a tubular passage with the object under test mounted in the middle. Air is made to move past the object by a powerful fan system or other means. The test object, often called a Wind Tunnel model, is instrumented with suitable sensors to measure aerodynamic forces, pressure distribution, or other aerodynamic-related characteristics. One of the most important parts of a Wind Tunnel is the flow visualization it provides. Sure lift, drag and efficiency can all be calculated with complex equations.

However, it is the visual aspect of a Wind Tunnel and the controllable environment it provides that allows you to physically see what will happen in multiple real life situations. You can create an environment where you can see how a plane will react when it is taking off, cruising and landing all in the confines of a test lab. Then, with the same machine, you can see how air flows over the body of a race car when it is zooming around a track to maximize its efficiency. The versatility and tangibility of a Wind Tunnel is what makes it such an important part of aerodynamic research.Small scale Wind Tunnels are fast becoming a significant research apparatus used in aerodynamic investigations to study the effects of air moving past solid objects.

2. REVIEW OF PAST RESEARCH

The flow separation development analysis in subsonic and transonic flow regime of the laminar airfoil

Robert placeck has done this study and found that The flow separation phenomenon is related to each aviation aero foil and is associated with the break-off of the thin layer (called boundary layer), right at the wing surface. The way in which the flow separation develops to the moment of the full separation occurrence, is strictly dependent on various factors: an aero foil thickness (thin, moderate, and thick), an airfoil type (turbulent, laminar, and supercritical), an aero foil surface quality (smooth or with roughness), the angle of attack, flow conditions (altitude and air turbulence), and Reynolds number. The exemplary and Wind Tunnel tests of a laminar aero foil have been performed at the Institute of Aviation in Warsaw. The main goal of the investigation was to study the separation process development in subsonic and early transonic flow regime. The aero foil chord was 0.2m. During Wind Tunnel test the natural laminar-turbulent transition was applied. The Mach numbers were 0.3 and 0.7. Reynolds number was approximately equal to 1.22.106 and 2.85.106 respectively. The angle of incidence was increased up until the flow was fully separated. During the experimental research, chosen test methods such as pressure measurements and schlieren visualization were applied. Wind Tunnel results were analyzed in terms of aerodynamic coefficients and flow separation type identification. The Wind Tunnel investigation revealed that separation phenomena at subsonic and transonic flow regime affected in a different manner on the airfoil aerodynamic performance. This was mainly because of the change of the flow pattern influencing on the separation process.

2. A Century of Wind Tunnels since Eiffel

Bruno Chanetz has found and presented that regarding the Wind Tunnel types and history fly higher, faster, preserve the life of test pilots and passengers, many challenges faced by man since the dawn of the twentieth century, with aviation pioneers. Contemporary of the first aerial exploits, Wind Tunnels, artificially recreating conditions encountered during the flight, have powerfully contributed to the progress of aeronautics. But the use of Wind Tunnels is not limited to aviation. The research for better performance, coupled with concern for energy saving, encourages manufacturers
of ground vehicles to perform aerodynamic tests. Buildings and bridge structures are also concerned.

studies for nearly a century. One area that remains of high importance is the reduction of the turbulence intensity across the Test Section. Today, there exists some design rules that have been determined through numerous experimental arrangements in addition to some theoretical guide lines; however, they are based on a considerable bed of assumptions. However, both types of guidelines may differ for particular Wind Tunnel applications. Finally, they have found that the trip wires will have the effect on the aero foil placed in the Wind Tunnel.

3. Design Procedure

The parameters that need to be defined in order to start the overall design are: Width (W_{TC}), Height (H_{TC}) and length (L_{TC}) These parameters allow computing the cross-sectional area: $\mathbf{S_{TC}= W_{TC}\, H_{TC}}$, and the hydraulic diameter: $\mathbf{D_{TC}=2\, W_{TC}\, H_{TC} / (W_{TC}+ H_{TC})}$.

- Contraction ratio, N≈5 for low quality flow, and N≈9 for high quality flow
- Maximum operating speed, V_{TC}.

According to the impact on the Wind Tunnel dimensions and flow quality, Classification of the design variables divided into two categories: main and secondary design parameters.

Main Design Parameters.

Maximum operating speed, V_{TC} Test chamber width, W_{TC} Test chamber height, H_{TC} Test chamber length, L_{TC}
Contraction ratio, N

Secondary Design Parameters.

Contraction semi angle, $\alpha_{C/2}$
Settling chamber length, l_{SC}
Diffuser semi angle, $\alpha_{D/2}$
Diffuser length, l_D

Aspect ratio, AR

Now, following the guidelines given above, such as the convergence angle and the contour line shape of the contraction zone, the test and contraction chamber can be fully defined. In the case when both opening angles, α and β, are the same, the contraction length, L_C, is given by the expression: $\mathbf{L_C= [(N)^{1/2} – 1]*W_{TC}/2*tan (\alpha_{C/2})}$.

Continuing in the upstream direction, the next part to be designed is the settling chamber. The only variable to be fixed is the length, because the section is identical to the wide section of the contraction. In the case when high quality flow is required, the minimum recommended non-dimensional length based on the hydraulic diameter, l_{SC}, is 0.60. This results from the necessity to provide extra space for the honeycomb and screens. In all other cases, the non-dimensional length may be 0.50. Therefore, the length of the settling, L_{SC}, chamber is given by: $\mathbf{L_{SC}= (N)^{1/2} *W_{TC} *l_{SC}}$.

Going downstream of the test chamber, we arrive at the Diffuser 1. Assuming that both semi opening angles are 3.5°, its non-dimensional length, l_{D1}, is the only design parameter. Although it has a direct effect on the Wind Tunnel overall length, we must be aware that this Diffuser together with corner. According to the experience, $l_{D1}>3$ and $l_{D1}>4$ is recommended for low and high contraction ratio Wind Tunnels respectively. The length of the Diffuser, L_{D1}, and the width in the wide end, W_{WD1}, is defined by: $\mathbf{L_{D1}=W_{TC} *l_{D1}}$ and $\mathbf{W_{WD1}= [1+2*l_{D1}*tan (\alpha_{D1/2})] *W_{TC}}$.

Therefore, we can already formulate the overall Wind Tunnel length, L_{WT}, as a function of the test chamber dimensions, the contraction ratio, and other secondary design parameters: $\mathbf{L_{WT} =L_{TC} +L_C+ L_{SC} + L_{D1}}$. This quick calculation allows the designer to check whether the available length is sufficient to fit the Wind Tunnel. Taking into account all the recommended values for the secondary design parameters, a guess value for the Wind Tunnel overall length, with a contraction ratio N=9 (high quality flow), is given by the formula: $\mathbf{L_{WT} =L_{TC} +16\cdot W_{TC}}$.

3.2.1 Design of Airfoils

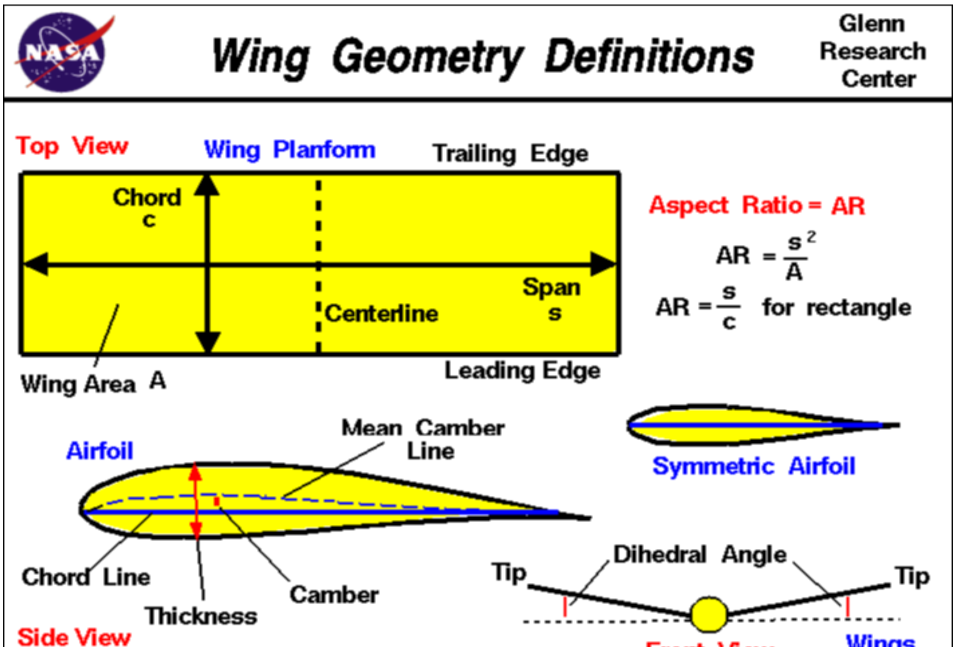


Fig 3.7 Wing Geometry

E231(e231-il) is the name of a particular Aerofoil profile, widely used in general purpose aircraft designs, and much studied in aerodynamics over the years. The profile was designed in 1922 by Virginius E. Clark. The aerofoil has a thickness of 11.7 percent and is flat on the lower surface from 30 percent of chord back. The flat bottom simplifies angle measurements on propellers and makes for easy construction of wings on a flat surface.

For many applications the E231(e231-il) has been an adequate airfoil section; it gives reasonable overall performance in respect of its lift-to-drag ratio, and has gentle and relatively benign stall characteristics. But the flat lower surface is not optimal from an aerodynamic perspective, and it is rarely used in modern designs. Aspect ratio is a measure of how long and slender a wing is from tip to tip.

The Aspect Ratio of a wing is defined to be the square of the span divided by the wing area and is given the symbol AR. For a rectangular wing, this reduces to the ratio of the span to the chord length as shown in fig 3.13.

$$AR = s^2 / A = s^2 / (s * c) = s / c$$

High aspect ratio wings have long spans (like high performance gliders), while low aspect ratio wings have either short spans (like the F-16 fighter) or thick chords (like the Space Shuttle).

3.3 Design Calculations
3.3.1 Test Section

The conditions at the Test Section are;

- Square inlet area ($230*230\ mm^2$)
- Test Section length (750mm)
- The dynamic pressure ($245\ N/m^2$) and velocity (45m/s)

From eq. $\delta=0.479*X/(R_X)^{1/2}$ the displacement thickness based on Reynolds number of (3.3*106) is calculated to be equal to (δ*=3.55 mm). So, the longitudinal direction with width constant along the Test Section becomes (7.01 mm), it can be approximated to (10 mm) so that the Test Section dimensions become:

- Test Section entrance→ ($230*230mm^2$)
- Test Section exit→ ($230*230mm^2$)
- Test Section length→ (750 mm)

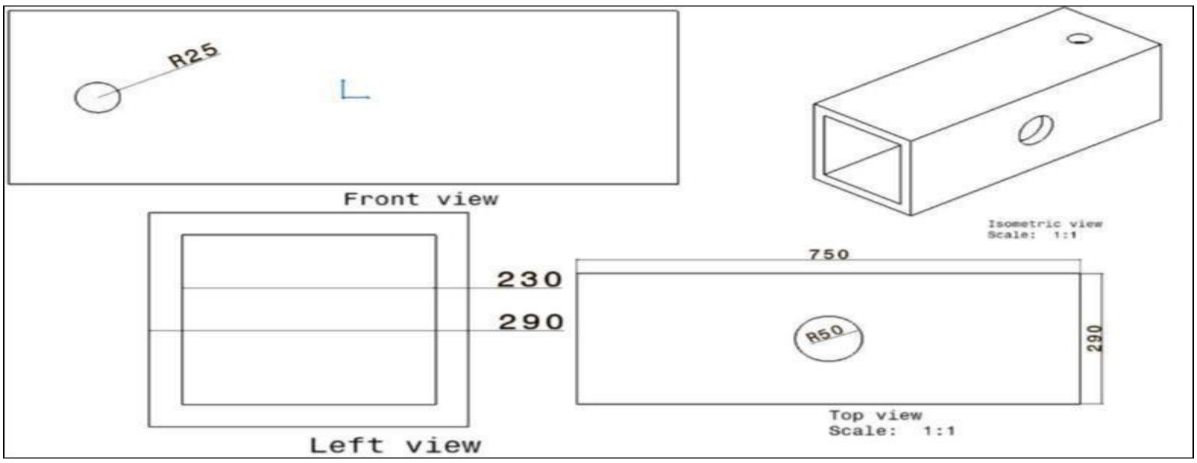


Fig 3.8 Test Section Line Diagram

The pressure losses occur at Test Section could be calculated by Assuming constant cross section and the friction factor also constant. So that, from eq. $K_{TS} = f * L/Do$ ($k_{TS} = 0.021$) for friction factor (f = 0.01).

3.3.2 Contraction Section

For the contraction ratio (6.16: 1) where inlet area is (570x570 mm^2) and exit area equal to (230x230 mm^2), the numerical solution was made with the following parameters-

Tunnel Nozzle Dimensions-

L =550 mm W$_1$= 570mm

H$_1$=570mm W$_2$= 230mm H$_2$=230mm

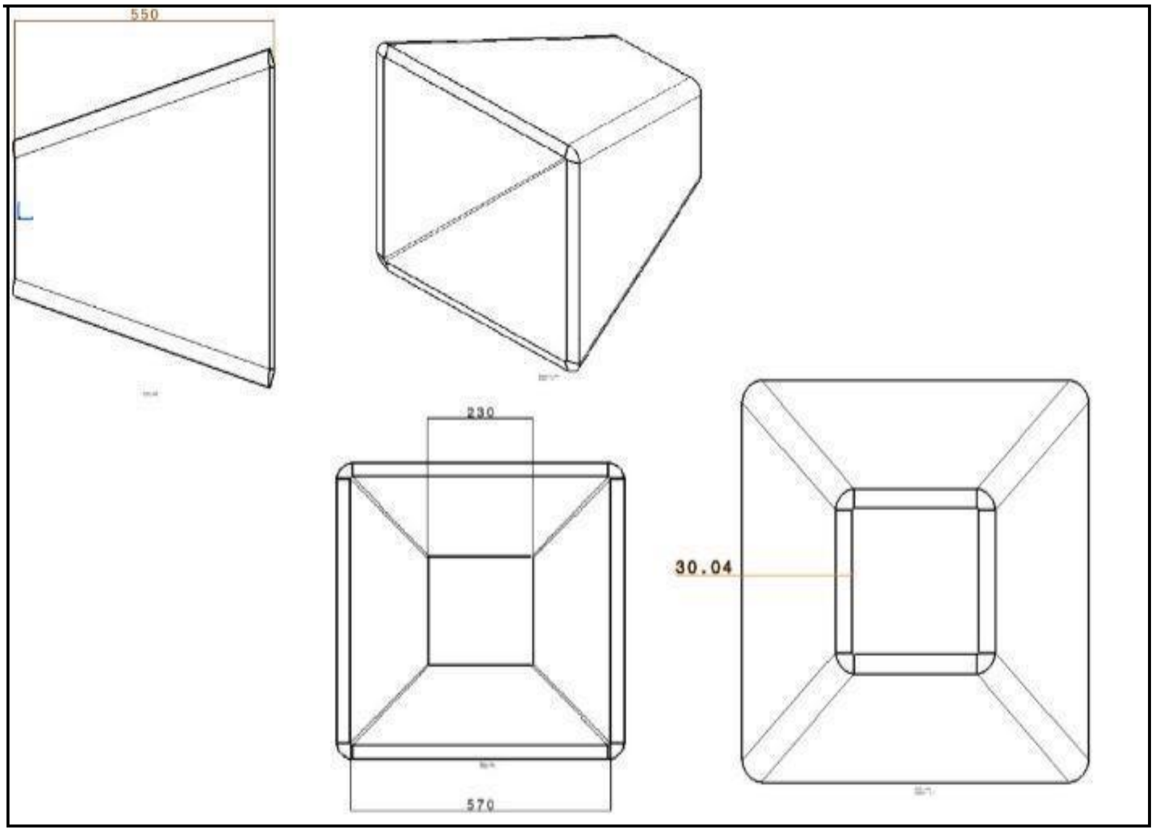


Fig 3.9 Effuser line Diagram

3.3.3 Diffuser

The energy losses at any point in the Wind Tunnel depend on the cubic velocity at that point. So that, the Diffuser works to decrease the velocity with minimum losses and higher back pressure. Generally, it must decrease the velocity without boundary layer separation at the wall. The following are the Diffuser dimensions:

- Inlet Diffuser Width W1= 230mm
- Outlet Diffuser diameter D2=450mm = fan diameter
- Diffuser length =100mm
- The divergence angle is = θ = 3^0

• Aspect ratio of Diffuser AR = 3.006:1

When comparing the Diffuser dimension with cone Diffuser, the ratio will be (N/R1) =4 this mean the divergence angle will be equal to 3.13◦ . So, it represents good approximation for dimensions.

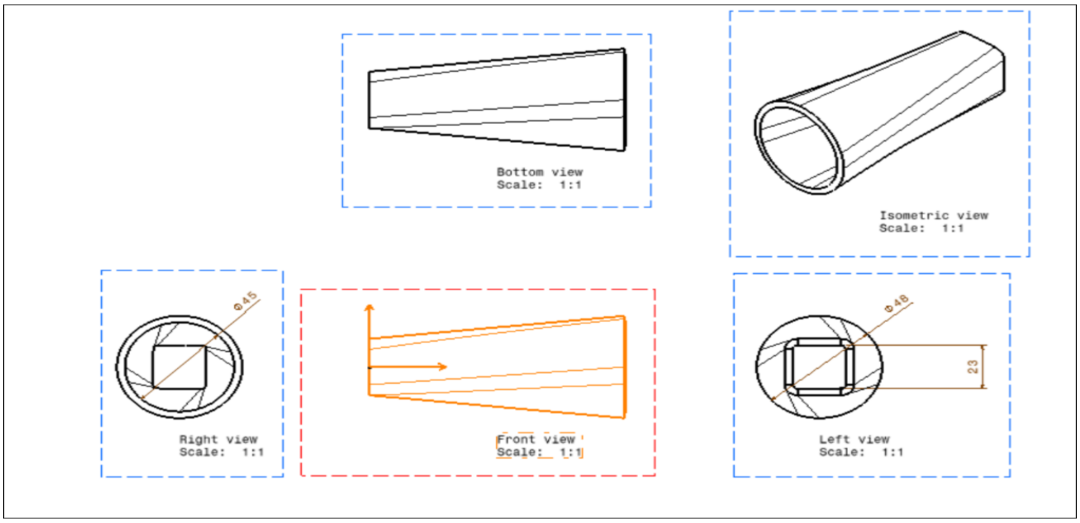


Fig 3.10 Diffuser Line Diagram

3.3.4 Settling Chamber and Straightener

The dimensions of settling chamber are,
• Length of section = 300 mm

• Entrance area (570x570) mm^2

3.3.5 Fan and Fan hub

A major work in designing fan is that it must provide a required velocity at Test Section (70 m/s) and to resist the decrease in pressure along the Wind Tunnel. Usually a safety factor may be taken as 25%.

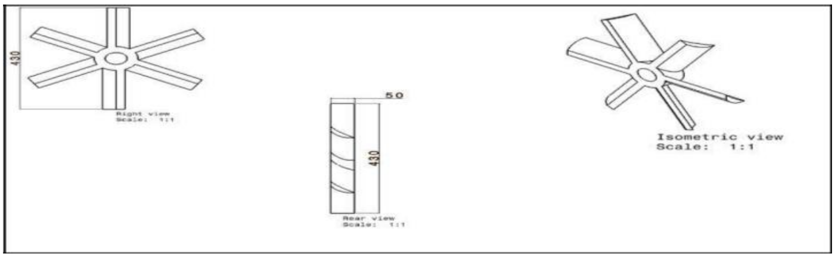


Fig 3.12 Fan

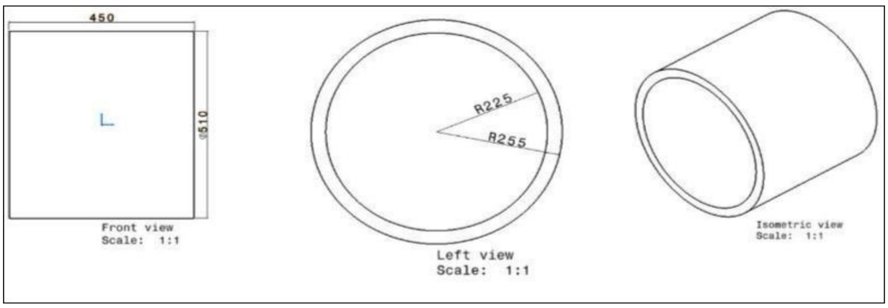


Fig 3.13 Fan hub

The following are dimensions and characteristics of fan

- Axial simple fan
- Hub diameter 45 mm
- Blade length is 450 mm
- Number of blades are 6
- Tilt angle of blade is 30^0
- Outer diameter of fan is 5100 mm
- Number of revolution per minute is 2380 rpm
- Steel sheets with thickness equal to 6mm are used in manufacturing the blades.

3.3.6 Airfoil

The dimensions of airfoil are,

Aspect Ratio=1.14:1 ,Chord=150mm, Span=170mm

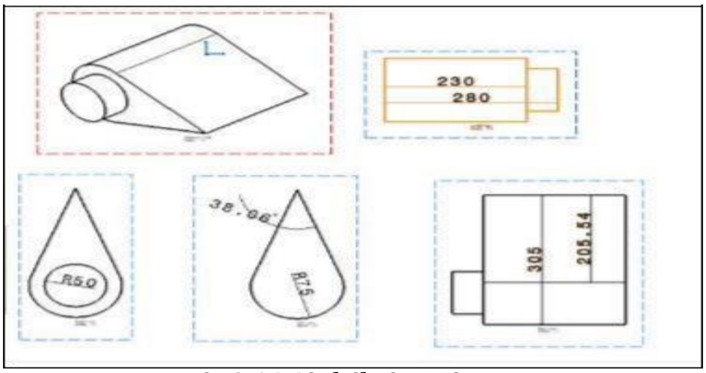


Fig 3.14 Airfoil Line Diagram

3.4 Design

The analysis of the components of Wind Tunnel has been done after designing them in CATIA software according design procedure. After analyzing the individual components in detail, they are fabricated

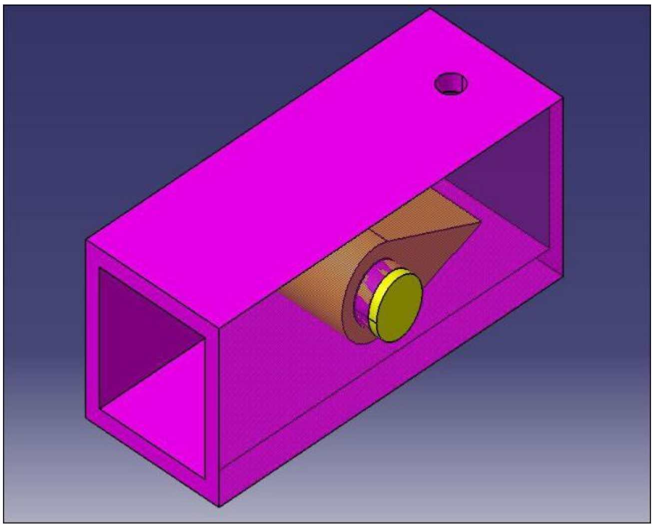


Fig.3.15 Test Section and Aerofoil

3D design of Test Section is created according to design dimensions by using PAD, POCKET, EXTRUDE tool in CATIA software. As CATIA is the good designing and analysis tools and easy to operate, we have chosen it.

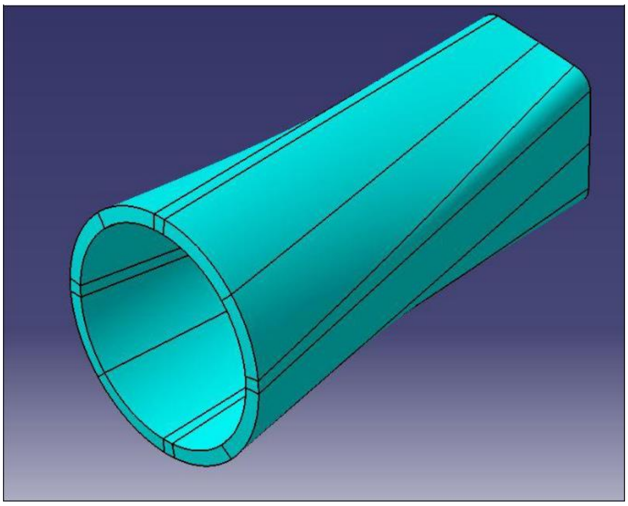


Fig.3.16 Diffuse

Diffuser with the length of 100mm and thickness 3mm is designed with help of Sheet Metal design tool.

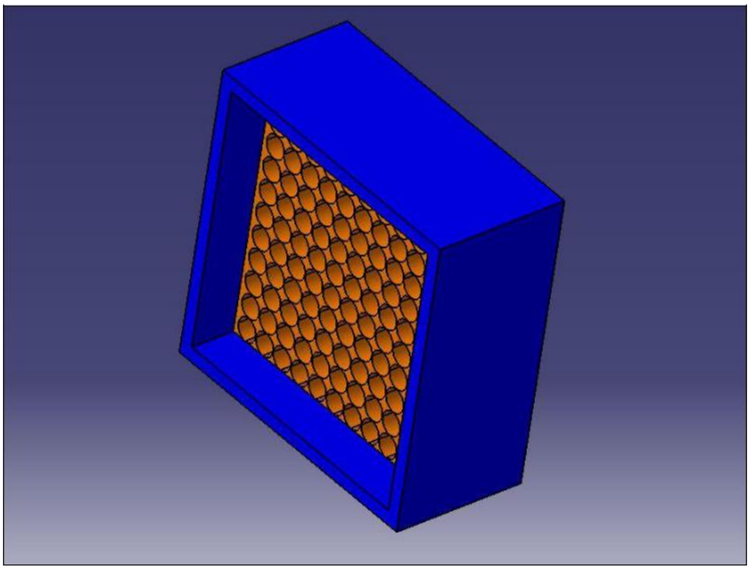


Fig.3.17 Settling chamber with honey comb structure

After Designing Settling Chamber and honey Comb structure, they are assembled together for the Analysis. Where each component is analyzed for the required parameters like pressure, velocity, temperature etc.

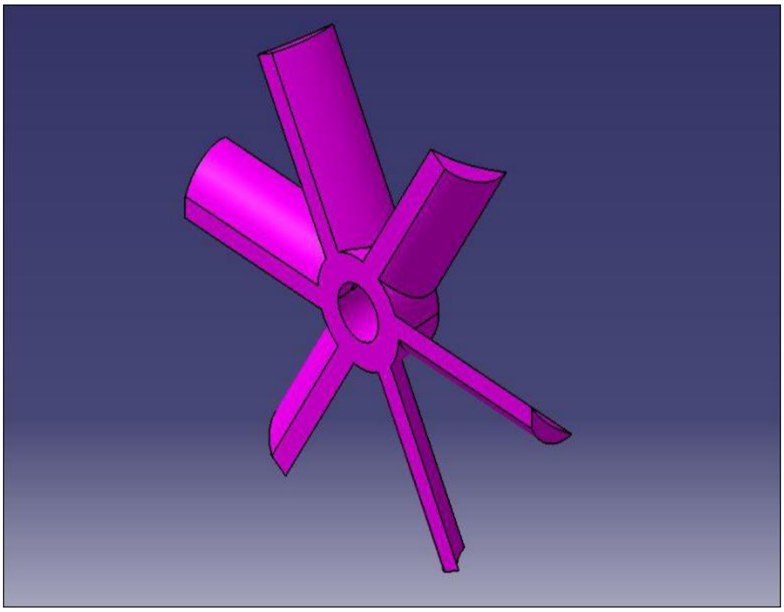


Fig.3.18 Fan

Finally, after designing each and every part of the Wind Tunnel they are assembled together to get the final required Wind Tunnel apparatus by using Assembly tools and the working model is analyzed in the ANSYS software.

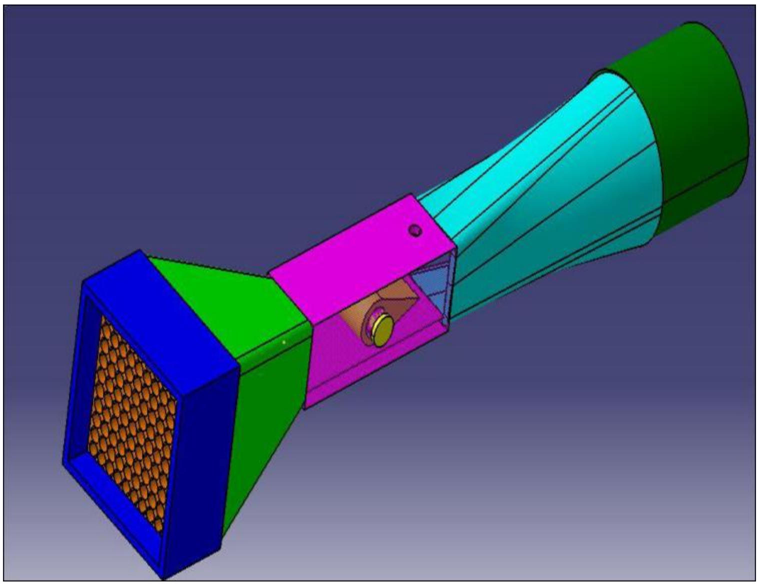


Fig.3.19 Assembly of Wind Tunnel

3. Testing & Analysis (CFD)

4.1 Working

Test object is kept in the testing chamber, by fixing it on a support. This support is in turn connected to the force measuring sensor (Strain Gauge). Suction fan is switched on which makes the smoke to pass over the object. Honeycomb provided at the entrance makes the flow laminar. Next both inlet and exhaust fans are switched on and air is allowed to pass over the test specimen. The lift generated by the specimen is noted in the Strain Gauge.

Next the lift generated is noted for various angles of attacks. Next the velocity of the air is varied using a fan speed regulator. The effect of velocity on aerofoil lift for different aerofoil models for various angle of attack is tabulated and then analyzed with the help of graphs.

4.2 Testing Bodies

There will be two airfoils used in this experiment airfoil CLARK Y. The airfoil has a chord length of 150mm and span of 170mm, which is equal to the width of the Test Section. In order to measure the pressure distribution on the airfoil surface, pressure taps were provided on each and every hole provided on airfoil section, as shown in Fig. The airfoil was mounted with the help of a frame inside the Test Section. With the help of a round protractor, the desired angle of attack for the airfoil was set. The airfoil was held at this angle using a screw mechanism. Measurements of surface pressure distribution were carried out with the help of water manometers to which all the tapping were connected. A slot was provided in the top wall of the Test Section for traversing the pitot tube to measure the velocity, at the desired location. The Test Section blockage was checked at the maximum angle of attack of 15^0. As the WIG craft fly at low angles of attack, most of the measurements such as those of lift and drag forces and velocit y survey over the airfoil were limited to this angle of attack. A maximum blockage ratio of about 6% was found.

4.4 Wind Tunnel Simulation

In recent years, ANSYS represents a good tool to find a mechanical and flow properties for complex configuration. Therefore, it was used to simulate the flow inside Wind Tunnel of the present work. The first step in simulation is to model the Wind Tunnel and to create its parts to apply the boundary conditions. These boundary conditions must be applied accurately to ensure good results. Figs in previous chapter show CATIA steps for the present Wind Tunnel (Modeling, Boundary condition, meshing). ANSYS solution shows that Test Section is approximately have a constant velocity near (70 m/s) along Test Section, which represents most important part of the Wind Tunnel. This indicate that our design is fair enough since there is no flow separation in velocity at Test Section or at least no thickening boundary layer at this region which may cause an error in measurements.

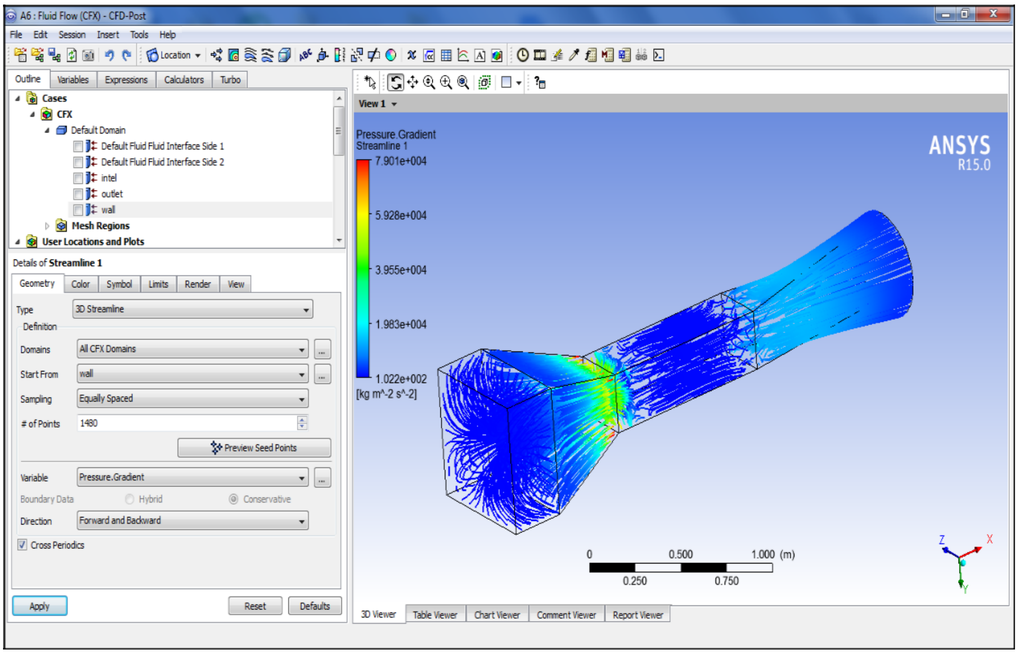


Fig.4.1 Pressure Contour

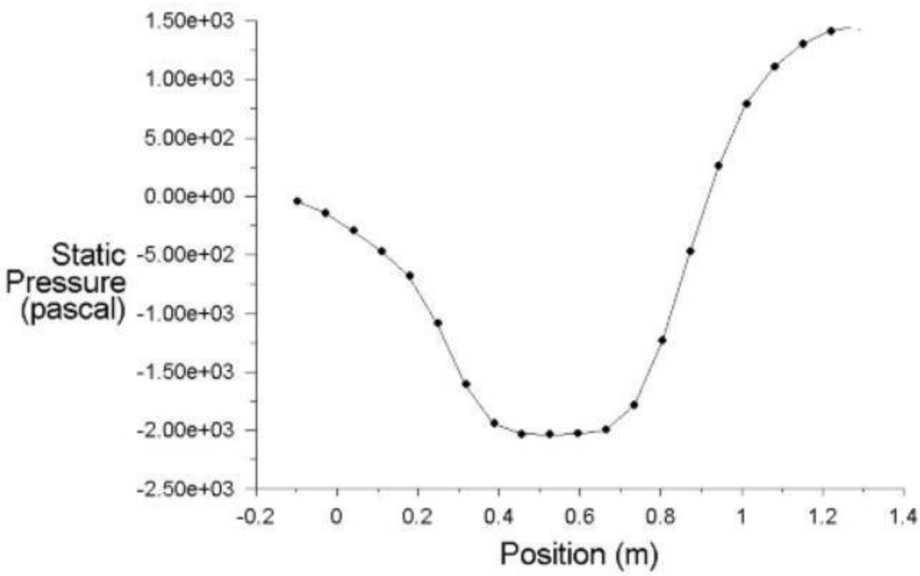


Fig.4.2 Static Pressures at different locations

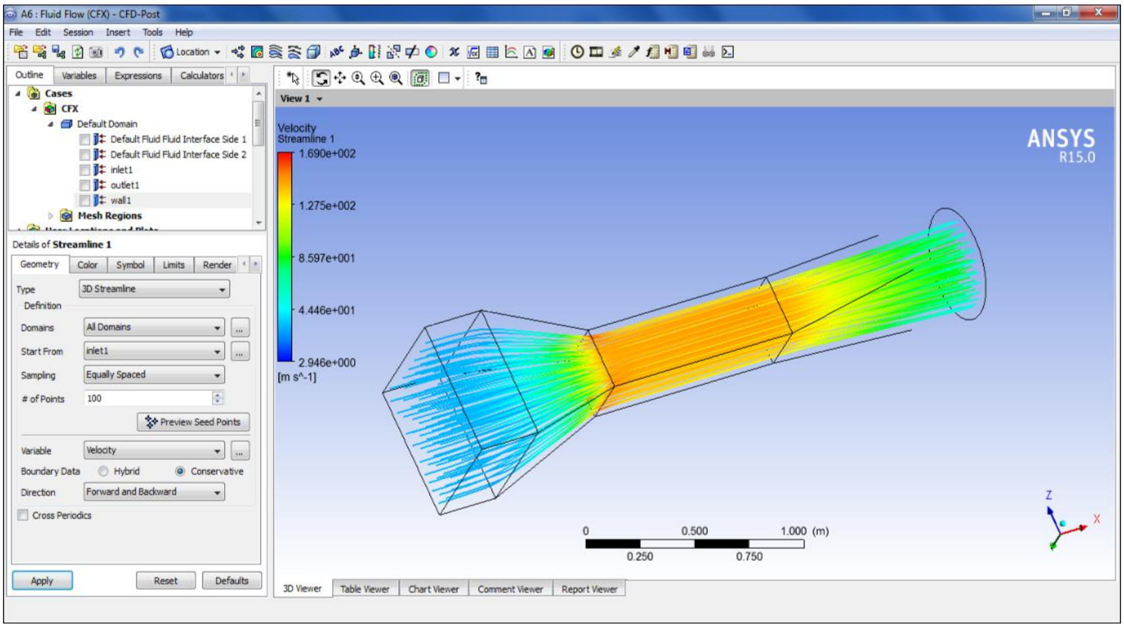


Fig.4.3 Velocity contour

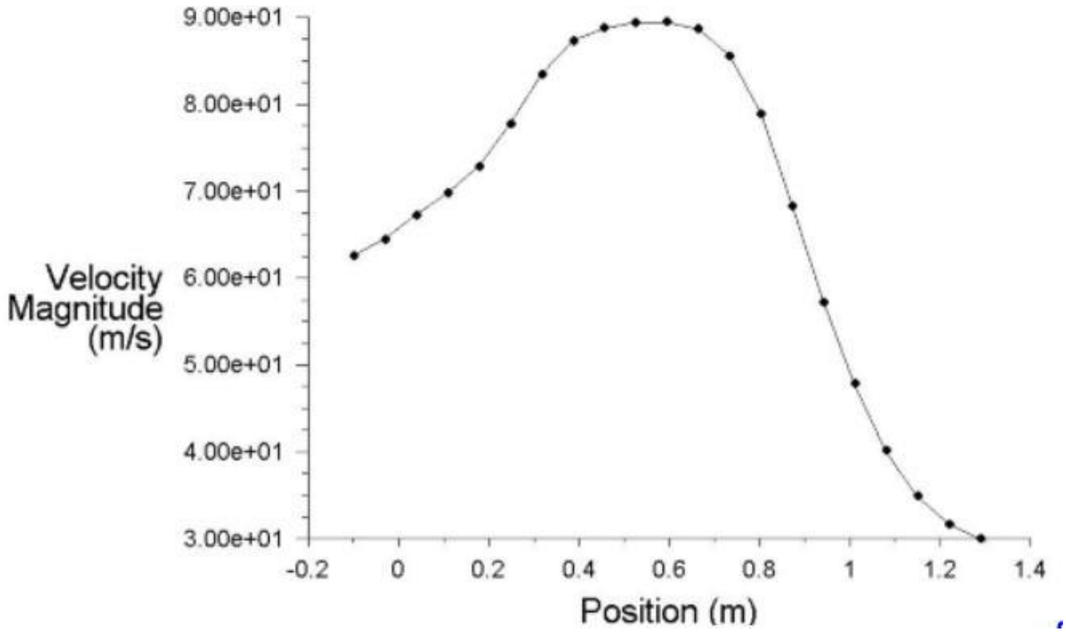


Fig.4.4 Velocity magnitude at different positions

4. RESULTS AND DISCUSSION

We have done CFD simulation on Clark Y Airfoil on Low Speed Open Type Wind Tunnel at constant velocity and data choosen are as follows. Free Stream Flow Velocity(V) = 20 m/s

Free Stream Dynamic Pressure ($Q\infty$) = 0.5*Density*V^2 =245 N/m^2 Chord Length(c) = 15 cm , Span Length(s) =17cm

Area of Airfoil(A) =0.15m*0.17m=0.0255 m^2

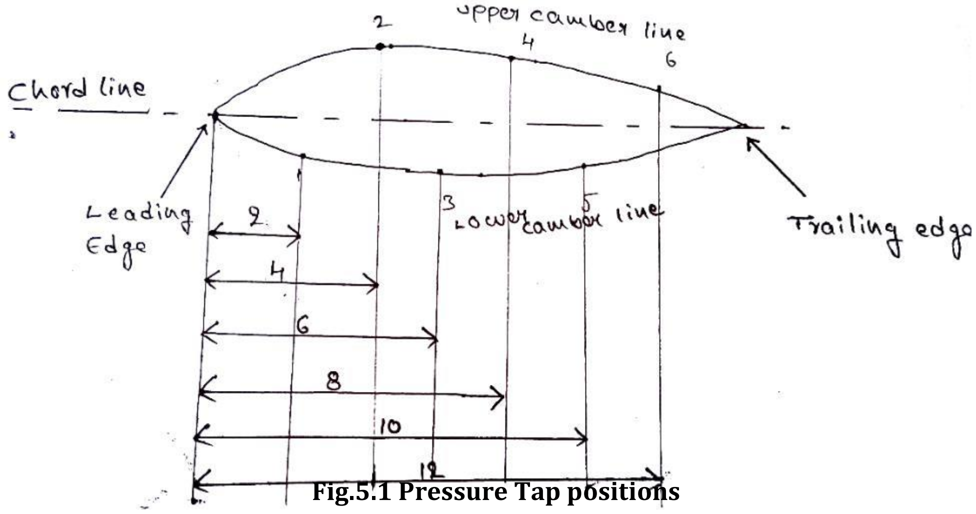


Fig.5.1 Pressure Tap positions

Table 5.1 Pressure Head Difference

Pressure Point/Angle of Attack(α)	Pressure Head Difference In Manometer (Δh)					
	1	2	3	4	5	6
15	27.8	5.1	9	5	6.2	6.3
10	17	6.6	11.5	6	7.2	7.3
5	15	7.6	11.1	6.5	7.2	7.5
0	14	9.6	11.6	7.9	8.4	8.1
-5	10.5	10.1	10	8	7.2	7.3
-10	9	10.1	9.5	8	7.6	6.8
-15	7	12.1	8	10	6.7	8.3

Model Calculation

For 10^0 Angle of Attack (for P_1)

CP= $\rho*g*\Delta h/q_\alpha$ =1000*9.8*17/245

=6.8 Where, q_α = _ ρV^2

$C_N = \sum (Cp_L - Cp_U)$ — = (14.28-7.96)*0.29Where, dx=4.37cm, C=15cm

=1.47

$C_L = C_N \cos\alpha$ =1.47*0.98

= 1.45

$C_D = C_N \sin\alpha$ =1.47*0.17

= 0.25

$F_L = 0.5 * C_L * \rho * V^2 * A$ =0.5*1.45*1.21*20^2*0.0255

=8.95 N

$F_D = 0.5 * C_D * \rho * V^2 * A$ =0.5*0.25*1.21*20^2*0.0255 = 1.54 N

For -15⁰ Angle of Attack (for P_1)

$CP = \rho * g * \Delta h / q_\alpha$ =1000*9.8*7/245

=2.8 Where, $q_\alpha = \frac{}{} \rho V^2$

$C_N = \sum (Cp_L - Cp_U)$ — = (21.7-30.4)*0.29 Where, dx=4.37cm, C=15cm

= -2.52

$C_L = C_N \cos\alpha$ = -2.52*0.96

= -2.41

$C_D = C_N \sin\alpha$ = -2.52*-0.25

= 0.63

$F_L = 0.5 * C_L * \rho * V^2 * A$ =0.5*-2.41*1.21*20^2*0.0255

=-14.87 N

$F_D = 0.5 * C_D * \rho * V^2 * A$ =0.5*0.63*1.21*20^2*0.0255

= 3.88 N

Where, C_P = Coefficient of Pressure

C_N= Coefficient of Normal Force

C_L= Coefficient of Lift

C_D= Coefficient of Drag

F_L= Lift Force

F_D= Drag Force

α= Angle of Attack

Table 5.2 Pressure Coefficient

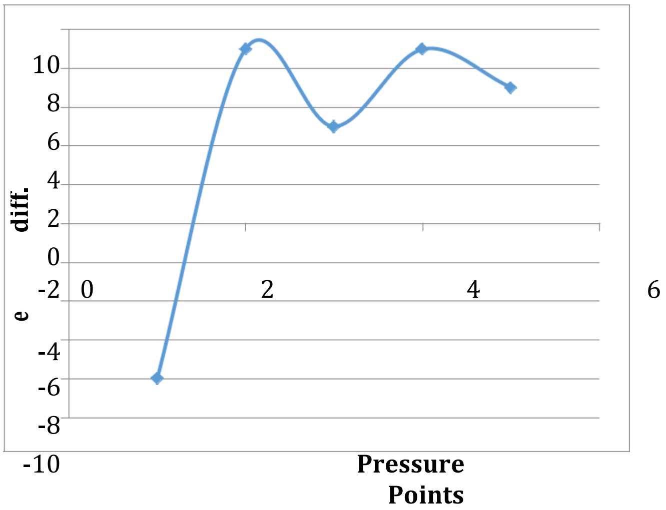
Angle of attack/Pressure Point	$C_P = \rho g \Delta h / q_\alpha$					
	1	2	3	4	5	6
15	11.12	2.04	3.6	2	2.48	2.52
10	6.8	2.64	4.6	2.4	2.88	2.92
5	6	3.04	4.44	2.6	2.88	3
0	5.6	3.76	4.6	3.16	3.36	3.24
-5	4.2	4.04	4	3.2	2.88	2.29
-10	3.6	4.04	3.8	3.2	3.04	2.72
-15	2.8	4.84	3.2	4	2.68	3.32

Table 5.3 Force Table

S.No	**α**	C_N	C_L	C_D	F_L	F_D
1	15	2.48	2.39	0.64	14.74	3.94
2	10	1.47	1.45	0.25	8.95	1.54
3	5	1.09	1.08	0.09	6.66	0.55
4	0	0.79	0.79	0	4.87	0
5	-5	0.83	0.82	-0.07	5.06	-0.43
6	-10	0.34	0.34	-0.06	2.09	-0.37
7	-15	-2.52	-2.41	0.63	-14.87	3.88

The flow characteristics over airfoil are studied by carrying out CFD simulation in a low speed wind tunnel. The pressure distribution on the airfoil surface was obtained, lift and drag forces were measured and mean velocity profiles were obtained over the surface. Simulations were carried out by varying the angle of attack, from -15^{0}to 15^0.

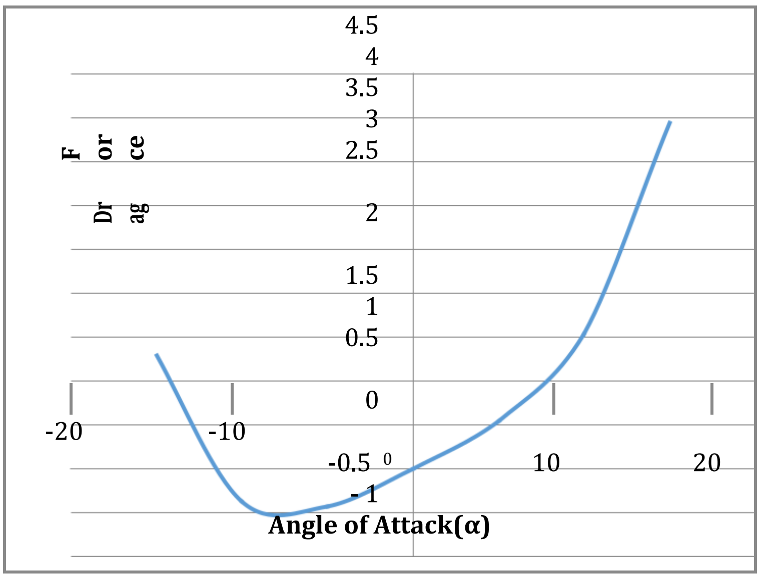
Plot 1: Pressure Points Vs Pressure diff.



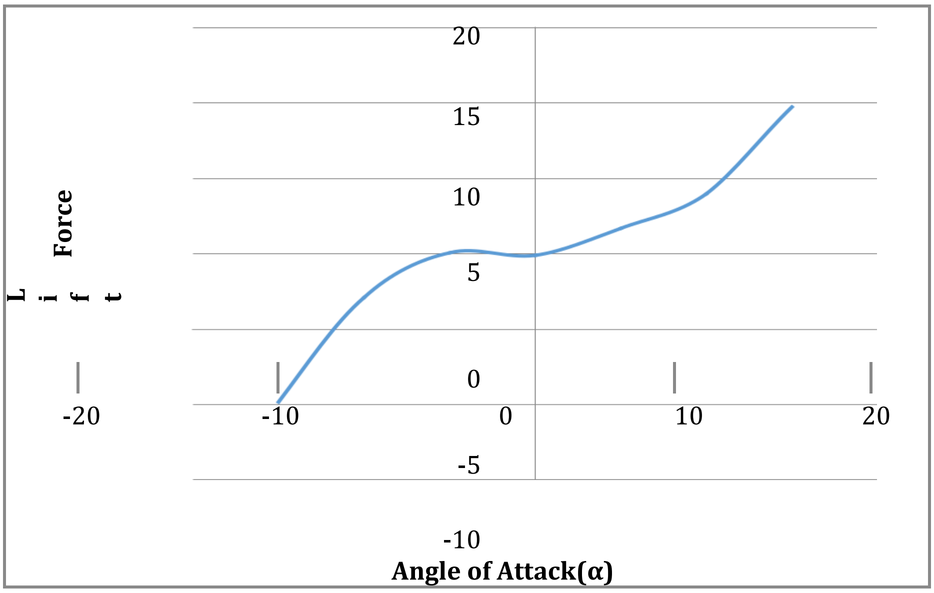
From the above graph, the variation of pressure at different pressure points from 1 to 6 is shown at a constant ground clearance with different angles of attack. From the first line i.e. series1 it is clearly shown that when the angle of attack is zero the pressure at points 1,3,5 are declining. This is because the pressure distribution on the top surface of the air foil is a streamline body and from just above the leading edge the velocity is increasing. We know from the Bernoulli's equation when velocity increases pressure decreases.

The nature of the pressure distribution curve and the normal force distribution curve is similar. The normal distribution curve is nothing but the lift force at each strip. It is the cosine component of the pressure force at each strip of the air foil. Angle of each pressure force to the vertical at each angle of attack is given in the table. From the first line i.e. series 1 it is clearly shown that when the angle of attack is zero the normal force from point1,3,5 gradually decreasing to zero. This is because the pressure distribution on the top surface of the air foil is gradually decreasing.

Plot 3: Angle of Attack (α) vs Drag Force



Plot 4: Angle of Attack (α) vs Lift Force



The lift force Vs the angle of attack graph shows the effect of lift force with increasing of angle of attack. Each line shows the variation of lift force with angle of attack at different ground clearance. When angle of attack is zero, the lift force is also zero for different ground clearance. The pressure force both the sides of the air foil are same. When we increase the angle of attack to 5 degree the lift force increases. But, further increasing of angle of attack, the lift force decreases. This point is called as stall point. Stalling angle we have found is around 15 degrees.

It was found that high values of pressure coefficient are obtained on the lower surface when the airfoil is close to the ground. This region of high pressure extended almost over the entire lower surface for higher angles of attack. As a result, higher values of lift coefficient are obtained when the airfoil is close to the ground. The flow accelerates over the airfoil due to flow diversion from the lower side, and a higher mean velocity is observed near the suction peak location. The pressure distribution on the upper surface did not change significantly with ground clearance for higher angles of attack. The upper surface suction causes an adverse pressure gradient specially for higher angles of attack, resulting in rapid decay of kinetic energy over the upper surface, leading to a thicker wake and higher turbulence level and hence a higher drag.

6. Conclusion

The review on the use of low speed Wind Tunnel applications shows that Wind Tunnels are very efficient for experimental simulations and flow visualization. The following conclusions can be drawn from the findings.

1. Designing of open type wind tunnel is carried out using CATIA V5 R20.
2. Analyzed the same wind tunnel by using ANSYS CFX software to check the functionality of the wind tunnel
3. Wind tunnel is designed for a specific purpose and speed range.
4. The pressure and velocity of the wind tunnel is determined at various positions and variations are plotted.
5. The streams lines are visualized for better understanding of flow simulation.
6. Hence it is recommended an open wind tunnel instead of a closed wind

References

[1] Robert Placek "The flow separation development analysis in subsonic and transonic flow regime of the laminar airfoil." A report on Transportation Research Procedia, Vol. 29, pp 323-329, 2018
[2] Bruno Chanetz "A century of wind tunnels since Eiffel" Volume 345, Issue 8, pp 581-594, 2017
[3] Kaveh Ghorbanian, Mohammad Reza Soltani "Experimental investigation on turbulence intensity reduction in subsonic wind tunnels." An Article on Aerospace Science and Technology, Vol 15, issue 2, pp 79-154, 2011

[4] Nguyen Quoc Y "Designing, Constructing, And Testing A Low – Speed Open – Jet Wind Tunnel www.ijera.com ISSN: 2248-9622, Vol. 4, Issue 1(Version 2), January 2014, pp.243-246

[5] Ishan.M.Shah, S. A. Thakkar, K. H. Thakkar, Bhavesh A. Patel "Performance Analysis on Airfoil Model in Wind Tunnel Testing Machine (WTTM)" (IJERA) ISSN: 2248-9622 www.ijera.com Vol. 3, Issue 4, Jul-Aug 2013, pp.2094-2103

[6] Md. Arifuzzaman, Mohammad Mashud "Design Construction and Performance Test of a Low Cost Subsonic Wind Tunnel" (IOSRJEN) e-ISSN: 2250-3021, p-ISSN: 2278-8719, www.iosrjen.org Volume 2, Issue 10 (October 2012), PP 83-92