

# Low-Speed Wind Tunnel design, optimization and CFD Analysis

Aniruddha Patil<sup>a</sup>, Aditya Manjarekar<sup>a</sup>, Mandar Aswale<sup>a</sup>, Saeed Nakade<sup>a</sup>

<sup>a</sup>Department of Mechanical Engineering, MIT World Peace University, Pune – 411038, Maharashtra, India

\*\*\*

**Abstract** - Wind tunnel is a tool used in aerodynamic research to study the effects of wind moving over the solid object. A typical wind tunnel consists of a motor and fan unit, settling chamber, contraction cone, test section and diffuser. The main work of the wind tunnel is to improve the design according to the aerodynamic shapes. In this project titled "Low Speed wind Tunnel Model" an effort has been made to make a detailed study on Wind tunnel design, CFD analysis of Wind tunnel. Our main goal is to list out correct references for designing any practically functioning Low speed wind tunnel. The calculations, the results table from research tables can also be used to manufacture a bigger wind tunnel well equipped with DAQ, which can be used in college and open to all students as lab equipment.

**Key Words:** Low Speed Wind Tunnel, Design Calculation, Optimization, CFD Analysis, Pressure and Velocity Cut Plots

## 1. INTRODUCTION

The practice of low-speed experimental aerodynamics has continued to evolve and it is a cornerstone in the development of a wide range of vehicles and other devices that must perform their functions in the face of forces imposed by strong flows of air or water. It was believed that in the 1970s and in the early 1980s the use of wind tunnels, especially in the subsonic regime, would rapidly disappear as computational fluid dynamics (CFD) would become a more attractive option to obtain data for many engineering applications, since it would be better in cost-effectiveness. Nevertheless, computational simulations improved since then but they have not come close to reaching a level sufficient to replace the need for experimental data in development projects.

Wind tunnels are used extensively to test the aerodynamic performance of aircraft and cars. Wind tunnels are large tubes with air blowing through it. The tunnels are used to replicate the actions of an object flying through the air or moving along the ground. As energy efficiency improvisation will be playing an important role in the future the need of wind tunnels is a must. It consists of three chambers i.e., contraction chamber, test chamber and diffuser chamber. Each one has its important role in the performance of wind tunnels and serve specific functions.

Low speed is referred to the air flow speed lower than 100 m/s, for which the incompressible flow condition is satisfied. The flow analysis over the model using low speed

air and studying different phenomena like boundary layer separation, turbulence, pressure recovery, drag, lift using experimental setup of wind tunnel is known as low-speed experimental aerodynamics. A well-designed wind tunnel could supply technical information for a large number of engineering applications such as external aerodynamics (flow over terrestrial and aerial vehicles), civil engineering (flow over bridges, buildings, cables, etc.), sport activities (flow over cyclists, design of volley and basket balls, wind sails etc.), fundamental fluid dynamics (laminar and turbulent flow over simple and complex geometries) and an extensive frame of other options in industry and research centers at universities around the world.

## 2. LITERATURE SURVEY

### J. B. Barlow, W. H. Rae [1]

The test section, which may be closed, open, partially open or convertible. The test-section-length-to-hydraulic-diameter ratio may typically be chosen to be 2 or more, in contrast to the shorter test sections of earlier era tunnels.

A diffuser of at least three or four test-section lengths. The typical equivalent cone angle is in the range of 2-33" with the smaller angles being more desirable. The area ratio is typically 2-3, again with the smaller values being more desirable.

Transition rectangular to circular cross section to take flow into the fan. Fan and straightener section. Other drive devices such as ejectors have also been used.

Wide-angle diffuser with separation control screens. Typical properties are angles of about 45" and area ratios of 24.

Contraction or node. Typical area ratings are in the range of 7-12, although lower and higher values are not uncommon.

### Bell J. H., Mehta R. D. [2]

The procedure of installing a contraction chamber consists of first computing the potential flow field and hence the pressure distributions along the walls of a contraction of given size and shape using a three-dimensional numerical panel method.

The pressure or velocity distributions are then fed into two-dimensional boundary layer codes to predict the behaviour of the boundary layers along the walls.

For small, low-speed contractions it is shown that the assumption of a laminar boundary layer originating from stagnation conditions at the contraction entry and remaining

laminar throughout passage through the successful designs if justified.

The measured boundary layer momentum thicknesses at the exit of four existing contractions, two of which were 3-D, were found to lie within 10 percent of the predicted values, with the predicted values generally lower.

### T. Gultop [3]

studied the impact of perspective degree on Airfoil performance. The reason for this study was to focus the ripple conditions not to be kept up throughout wind tunnel tests.

These studies indicate that aeroelastic insecurities for the changing arrangements acknowledged showed up at Mach number 0.55, which was higher than the wind tunnel Mach number point of confinement velocity of 0.3.

### P. Bradshaw, R.C.Pankhurst [4]

aerodynamic and structural design was studied from the viewpoint of the prospective tunnel designer, and details given of present-day practice. Comments, mostly cautionary, were made on the features of certain existing tunnels, and drawings and tabulated data were given for about thirty typical tunnels of various ages. The emphasis was on low-speed tunnels of wooden construction, but much of the discussion was applicable to the design of tunnels of larger size or higher speed.

### Nelton Chwee Yang Koo [5]

studied low speed wind tunnel design. This paper covered the process of design and fabrication of the small wind tunnel. A computer aided drawing (CAD) called Solid Work is use to design the wind tunnel. The drag force on a sphere in an air stream was measured at various free stream velocities below 100 m/s. This was done in a low-speed wind tunnel using an integral balance system to measure the drag force.

### Nagendra Kumar Maurya [6]

in this research work, low velocity wind tunnel was designed. It was concluded that shape, size, material selection and manufacturing process is very important for designing of wind tunnel.

Results depicts that velocity was increased and pressure was decreased when object height was gradually increases along the length up-to the maximum height and after the maximum height pressure was increased and velocity was decreases when height of object was decreases gradually along the length of object.

## 3. DESIGN METHODOLOGY

The main objective of this study was to design closed circuit low speed wind tunnel. A typical closed circuit wind tunnel consists of various section viz., axial flow fan, test section, diffuser, contraction chamber, etc. The details of the component of wind tunnel are discussed in next sub section.

### 3.1 Test Section Design

The wind tunnel design starts with deciding the test section keeping an eye on the accessibility and installation of the test model and instrumentation. The test chamber length has to be in the range of 0.5-3 times its hydraulic diameter. The test will be carried out at a flow speed of 5-7 m/s. The pressure loss coefficient goes on increasing with increase in the test section length. So, the length of test section should be as small as possible.

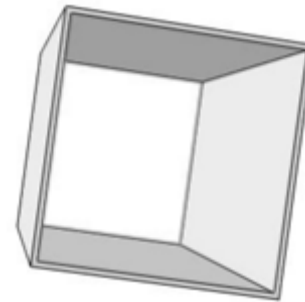


Fig- 1: Test Section

By considering availability of space, we have taken cross sectional area as 200mm × 200mm. So, hydraulic diameter will be 200mm. According to Barlow, chamber length should be in the range 0.5 -3 times hydraulic diameter and optimum for 2. Hence length of test section will be 400mm, hence test section dimensions are 400mm × 200mm × 200mm.

### 3.2 Diffuser Design

The diffuser is mainly used to reduce the velocity of flow in the shortest possible distance to reduce the load on drive system. The flow through the diffuser depends on its geometry defined by area ratio (ratio of outlet area to the inlet area) and diffuser angle ( $2\theta$ ), wall contour and diffuser cross-sectional shapes.

The diffuser connects the inlet section and the settling chamber. It is required that there is no flow separation inside this section. The area ratio between the inlet section and the outlet (of the settling chamber) of the diffuser is 2. It is suggested that the maximum diffuser angle suitable for that ratio to prevent flow separations is 4 deg.

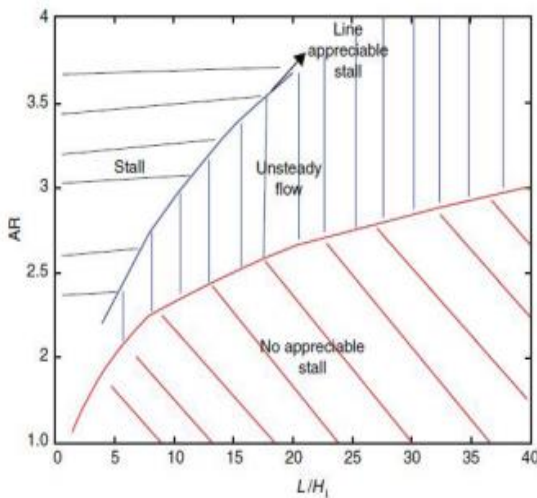


Fig-2: A diagram showing the diffuser

According to Kline’s flat diffuser curves, the diffuser should not produce any stall. Assuming minimum diffusion angle i.e., 4 deg and solving for L, the maximum of the minimum length of the diffuser is found to be 900mm. Hence diffusing section length 900mm is chosen.

### 3.3 Contraction Chamber Design

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. For smaller tunnels the contraction ratio should be in between 6 to 9.

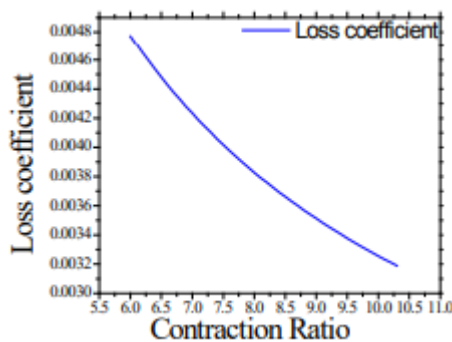


Fig-3: Contraction section pressure loss coefficient with contraction ratio

The pressure drop coefficient values go on decreasing with increase in CR, so the maximum allowable contraction ratio 7 is chosen. Area ratio should be 7. Also, the angle of contraction should be 32 degrees for best Accelerated flow. So, after calculations a contraction of length of 530mm is used. Hence entry dimension of contraction chamber is 530mm × 530mm.

## 4. BOUNDARY CONDITIONS

The power required to maintain steady flow through the wind tunnel is equal to the total losses occurring in the flow through the tunnel

These losses are due to kinetic energy being dissipated by vortices and turbulence. The loss in kinetic energy, which appears as a decrease in total pressure must be compensated by pressure rise provided by fan. Hence, fan selection is a very important thing which needs to be considered.

First step to perform CFD analysis is to set some boundary conditions. These boundary conditions are such like flow rate, atmospheric pressure, ambient temperature, etc.

Table-1: Different fans considered

Brand	Model	Flowrate (cfm)	Speed (rpm)	Sweep (mm)	Power (W)	Cost (Rs)
Usha	Crisp air LX 250mm	478	1250	250	45	1580
Usha	Striker high speed	450	2100	200	60	1760
Crompton	Flux air	380	1380	200	31	1370
Bajaj	Maxima Dxl	400	1150	250	34	1360
Bajaj	Freshee	380	1400	200	46	1591

We considered fans like USHA Crisp Air LX 250mm, USHA Striker High Speed, Crompton Flux Air, Bajaj Maxima Dxl, Bajaj Freshee. We chose USHA Striker High Speed fan which has flow rate of 450 cfm and 200mm sweep. It was because it had sweep of 200 mm and outer dimension of 295mm, which could help us in reducing wind tunnel’s overall size as well as cost.

Hence while feeding the boundary conditions in the softwares, we have used conditions such as atmospheric pressure at inlet of the contraction chamber i.e. 0 pa gauge, outlet volume flow rate of 450 cfm at the outlet of diffuser at the position of fan and ambient temperature of 298 K i.e. 25°C.

## 5. CFD ANALYSIS IN SOLIDWORKS

So far, for Test Section Design, we referred Barlow’s paper. For Contraction Chamber, we referred Bell & Mehta’s paper. For Diffuser, we referred Kline’s flat curves for deciding the length of our test section.

After studying different research papers, we came to know that, pressure drop is very important parameter in design, because lesser the pressure drop, lesser will be the losses. So, for finalizing the dimensions, here are some different iterations in which we are varying some parameters to get minimum pressure drop and accurate results.

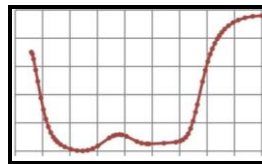
**1<sup>st</sup> Iteration:**

Name of section	Dimension	Area ratio	Angle of divergence(deg)	Pressure drop(Pa)
Test section	200 * 200 * 400 mm	1	-	5.89
contraction	500 mm length	6.69	32	
Diffuser	800 mm length	2.57	8	

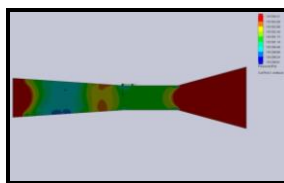
So, for the 1<sup>st</sup> iteration, test section is of 200×200×400 mm dimension. Contraction chamber is of 500 mm length and 5° angle of divergence. After the flow analysis, we got to know the pressure drop as 5.89 Pa. Observing the pressure and velocity graphs, we obtained that their variation is non-linear in this case.



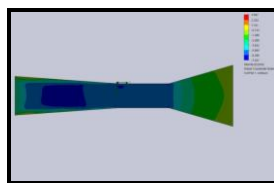
XY Plot - Velocity



XY Plot - Pressure



Cut Plots - Pressure



Cut Plots - Velocity

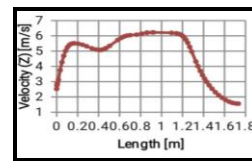
For wind tunnel, test section should have maximum velocity and minimum pressure but, in this iteration, test section is not having maximum velocity. Also, there are many irregularities which should not be there. We observed that, in diffuser section, the variation of pressure and velocity is abrupt.

After test section, we want velocity to decrease and pressure to increase in gradual manner, but this is not the case in this iteration. So, in the next iteration, we will change the dimension and try to minimize pressure drop and perform CFD analysis.

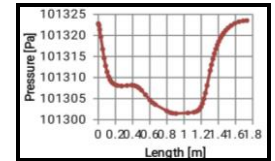
**2<sup>nd</sup> Iteration:**

Name of section	Dimension	Area ratio	Angle of divergence(deg)	Pressure drop(Pa)
Test section	200 * 200 * 400 mm	1	-	5.49
contraction	500 mm length	5.20	26	
Diffuser	850 mm length	2.52	7	

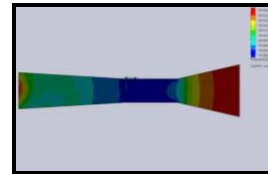
In 2<sup>nd</sup> iteration, we are reducing the divergence angle of contraction to 26° and increasing the length of diffuser by 50 mm and angle to 7°. After CFD analysis, we got the pressure drop as 5.49 Pa which is less than 1<sup>st</sup> iteration.



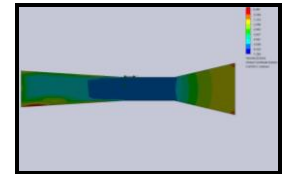
XY Plot - Velocity



XY Plot - Pressure



Cut Plots - Pressure



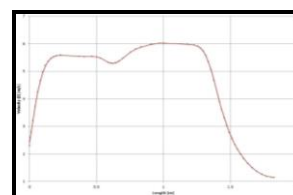
Cut Plots - Velocity

But pressure variation is not linear in this iteration. As earlier said, we want test section of maximum velocity and minimum pressure and we are getting maximum velocity and minimum pressure in this iteration. Also, diffuser should have gradual variation of pressure and velocity. But there are losses near the exit of the diffuser, hence we observed the variation in graphs. So, in next iteration, we will change the dimension and observe the result.

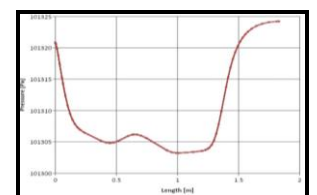
**3<sup>rd</sup> Iteration:**

Name of section	Dimension	Area ratio	Angle of divergence(deg)	Pressure drop(Pa)
Test section	200 * 200 * 400 mm	1	-	4.75
contraction	500 mm length	6.99	36	
Diffuser	950 mm length	2.49	7	

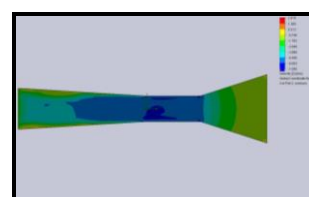
In 3<sup>rd</sup> iteration, divergence angle of contraction is 36° and length of diffuser is increased by 100 mm. After performing CFD analysis, we got the pressure drop as 4.75 Pa which is lesser than earlier cases.



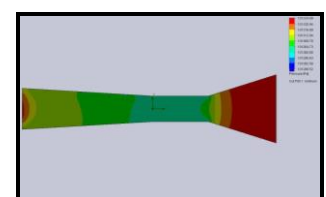
XY Plot - Velocity



XY Plot - Pressure



Cut Plots - Pressure



Cut Plots - Velocity

In this iteration, we observed that pressure and velocity cut plots are better than earlier cases. Contraction

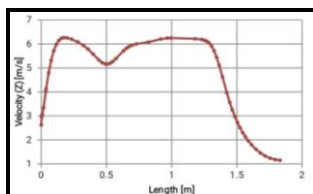


angle of  $36^\circ$  is giving better accelerated flow than earlier case. But there are still some irregularities that we don't want, it can be observed by pressure and velocity graphs. Pressure drop is lesser than earlier case which states that losses have decreased. So, in next iteration, we will change some parameters and perform CFD analysis again.

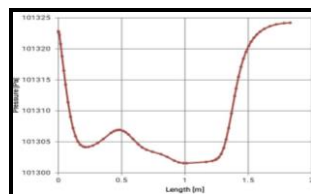
**4<sup>th</sup> Iteration:**

Name of section	Dimension	Area ratio	Angle of divergence(deg)	Pressure drop(Pa)
Test section	200 * 200 * 400 mm	1		4.59
contraction	500 mm length	6.99	36	
Diffuser	950 mm length	2.21	6	

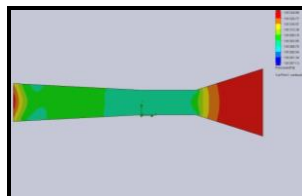
In 4<sup>th</sup> iteration, diffuser angle is reduced to  $6^\circ$ . After performing CFD analysis, we got pressure drop as 4.55 Pa which is lesser than earlier cases.



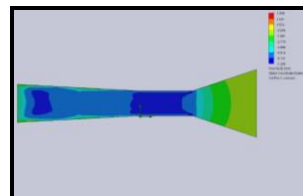
XY Plot - Velocity



XY Plot - Pressure



Cut Plots - Pressure



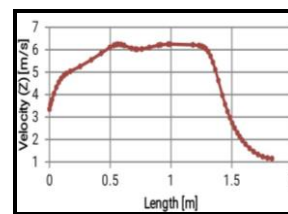
Cut Plots - Velocity

Again, there is abrupt variation in pressure and velocity plots. After observing cut plots, we came to know that velocity is maximum at 2 points, at the test section and behind the outlet of diffuser. But we want gradual variation in graphs, so in next iteration, we will try to eliminate that variation and minimize the pressure drop.

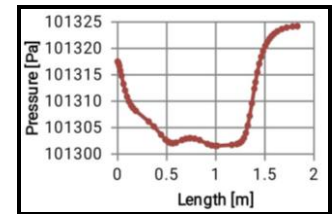
**5<sup>th</sup> Iteration:**

Name of section	Dimension	Area ratio	Angle of divergence(deg)	Pressure drop(Pa)
Test section	200 * 200 * 400 mm	1	-	4.13
contraction	500 mm length	6.99	36	
Diffuser	950 mm length	1.91	4	

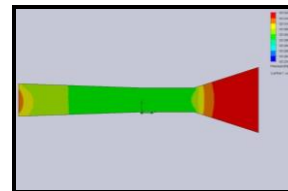
In 5<sup>th</sup> iteration, angle of divergence and diffuser is reduced to  $4^\circ$ . After performing CFD analysis, we got pressure drop as 4.13 Pa, which is lesser than earlier cases.



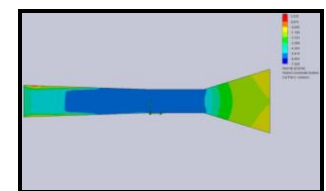
XY Plot - Velocity



XY Plot - Pressure



Cut Plots - Pressure



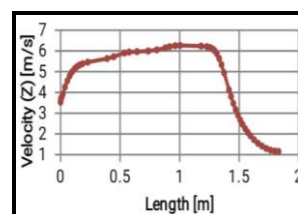
Cut Plots - Velocity

By observing cut plots, we came to know that velocity and pressure is constant in test section which is favorable condition. After that, pressure is increasing and velocity is decreasing. Still there is some boundary layer separation near the exit of the diffuser. So, we will try to eliminate this in further iteration.

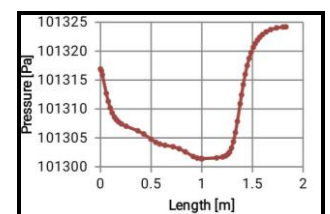
**6<sup>th</sup> Iteration:**

Name of section	Dimension	Area ratio	Angle of divergence(deg)	Pressure drop(Pa)
Test section	200 * 200 * 400 mm	1		3.81
contraction	500 mm length	6.99	36	
Diffuser	950 mm length	1.81	3.6	

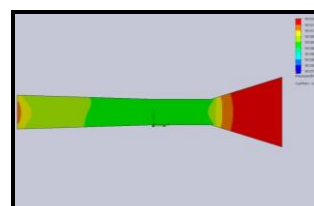
In 6<sup>th</sup> iteration, angle of divergence and diffuser is reduced to  $3.6^\circ$ . After CFD analysis, we got pressure drop as 3.81 Pa which is less than earlier cases.



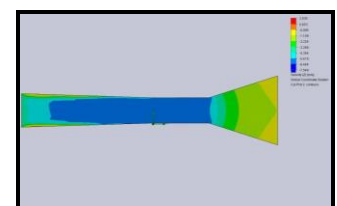
XY Plot - Velocity



XY Plot - Pressure



Cut Plots - Pressure



Cut Plots - Velocity

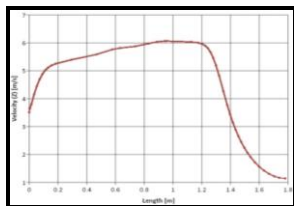
In this iteration, we observed that velocity and pressure is constant in test section. By reducing the diffuser angle, we got the reduction in pressure drop. The cut plots

show linear variation, but there is slight boundary layer separation near the exit of the diffuser. After referring research papers, we found that reducing the length of diffuser can help in eliminating boundary layer separation. So, in next iteration, we will reduce the length of diffuser and will perform CFD analysis.

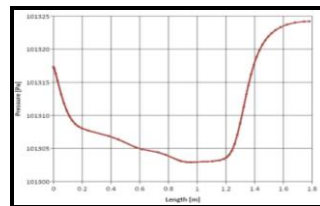
**7<sup>th</sup> Iteration:**

Name of section	Dimension	Area ratio	Angle of divergence(deg)	Pressure drop(Pa)
Test section	200 * 200 * 400 mm	1		3.5146
contraction	500 mm length	6.99	36	
Diffuser	900 mm length	1.64	5	

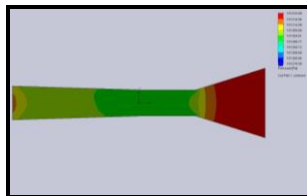
In 7<sup>th</sup> iteration, we are reducing the length of diffuser by 50 mm and reducing diffuser angle by 5°. After performing CFD analysis, we got the pressure drop as 3.51 Pa which is less than earlier iteration, so losses will be lesser in this case.



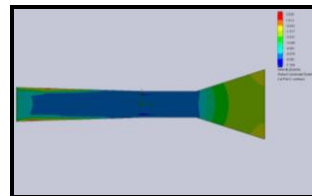
XY Plot - Velocity



XY Plot - Pressure



Cut Plots - Pressure



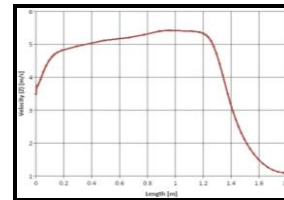
Cut Plots - Velocity

In this iteration, velocity and pressure graph are improved in terms of linearity. Test section is having maximum velocity. By decreasing the length of the diffuser, boundary layer separation at the exit of the diffuser is eliminated. These observations tell us that we are near to our solution. In next iteration, we will try to minimize pressure drop.

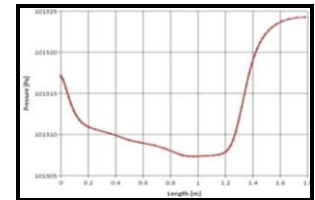
**8<sup>th</sup> Iteration:**

Name of section	Dimension	Area ratio	Angle of divergence(deg)	Pressure drop(Pa)
Test section	200 * 200 * 400 mm	1		2.48
contraction	500 mm length	6.99	36	
Diffuser	900 mm length	1.49	4	

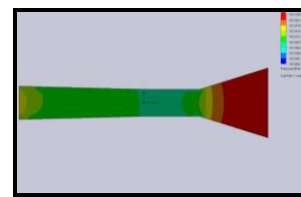
In 8<sup>th</sup> iteration, we reduced diffuser angle to 3°. After performing CFD analysis, we got the pressure drop as 2.48 Pa which is lowest among all iterations.



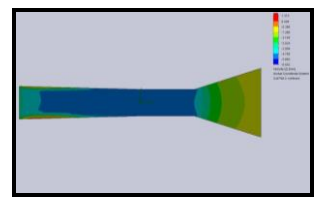
XY Plot - Velocity



XY Plot - Pressure



Cut Plots - Pressure



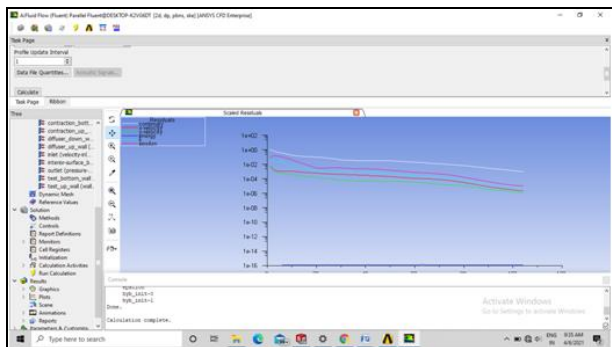
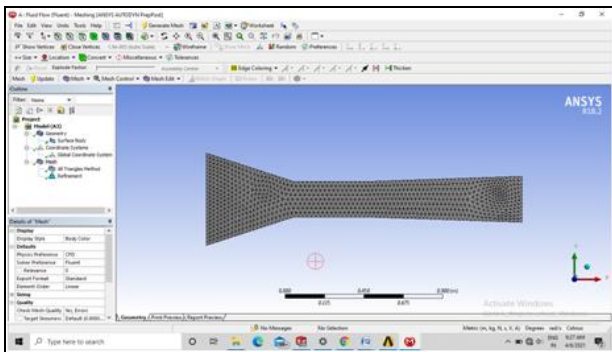
Cut Plots - Velocity

We got maximum velocity at test section. Pressure variation is linear across the wind tunnel than the earlier cases. Pressure drop across wind tunnel is very important parameter. This is the lowest pressure drop we achieved so far. We got to know that reducing the diffuser angle helps in the pressure recovery but we cannot further reduce the diffuser angle because it will be difficult for us to fit fan in such a small space. Also, further reduction in angle leads to sudden pressure variation which we don't want.

In our analysis, we have taken different contraction angles like 26°, 32° and 36°. Contraction angle of 36° gave us the best accelerated flow so far with area ratio 7. Diffuser angle of 4° have resulted us the minimal losses and flow uniformity. Hence with this design we will proceed further for our final CFD analysis in ANSYS fluent as it provides more accurate results and is best in practice.

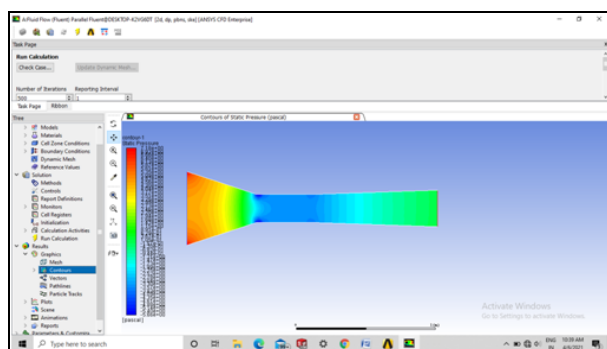
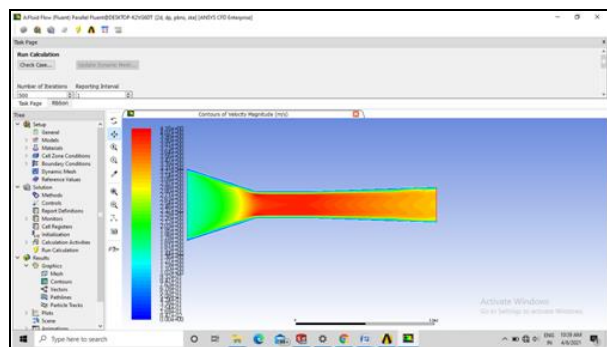
**6. MESHING AND SOLVER MODEL**

ANSYS represents a good tool to find a mechanical and flow properties for complex configuration. For wind tunnel meshing we used triangular mesh with refinement as it is simple and gives fast results. For the mathematical solver we used K Epsilon turbulence model which is the most common model used in CFD to simulate mean flow characteristics for turbulence flow conditions. We have used this model with energy equation which gave us very accurate plots.

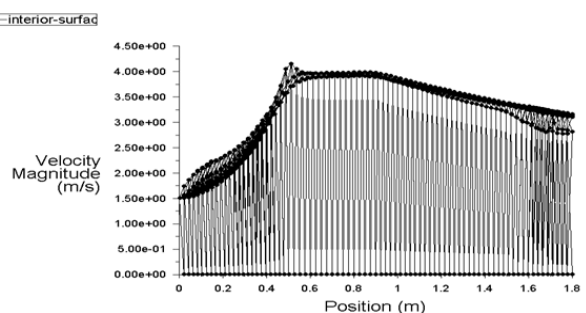
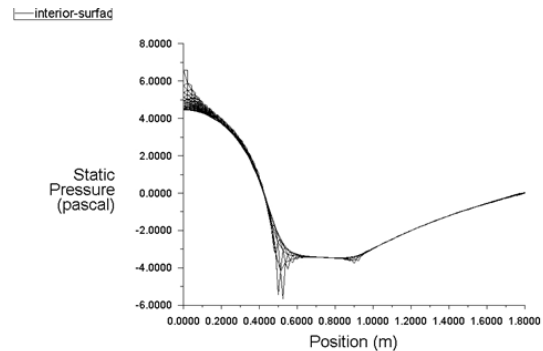


After drawing a wind tunnel in design modeler, proper meshing is done using mesh control feature in ANSYS. Then we gave boundary conditions same as that were decided earlier and calculated pressure-velocity contour cut plots as well as pressure-velocity graphs. We compared those results with the results we got from Solidworks flow.

### 7. ANSYS RESULTS



Cut plots and XY graphs obtained from ANSYS were more accurate than SolidWorks Flow Simulation because it takes into account the zero slip condition at boundary walls and turbulence tolerance.



The nature of pressure- velocity graphs obtained from ANSYS are bit different from solidworks in terms of number of fluid particles considered. The solidworks pressure-velocity graph is a single line as it considers only one particle but the ANSYS considers ten different particles at different differential time intervals hence the graph is combination of ten lines.

### 7. Modelling of designed wind tunnel

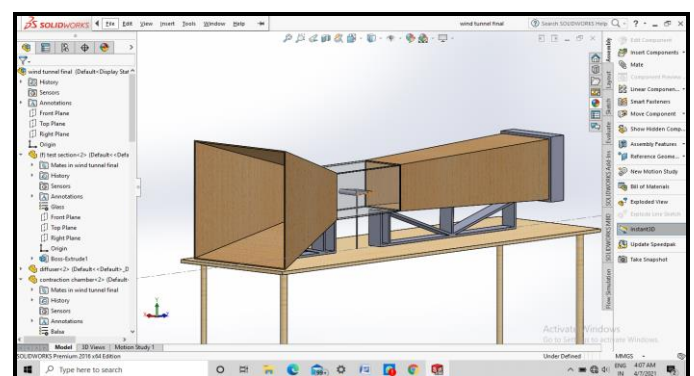
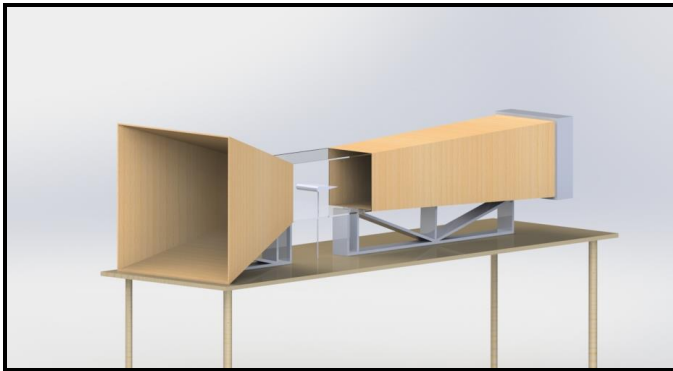


Fig-5: SolidWorks Modelling



**Fig-6:** Rendered Image

## 8. CONCLUSIONS

In this research work, the low-speed wind tunnel components such as the Test section, Contraction nozzle and Diffuser are designed and modeled in Solidworks using suitable design procedures as per the literature survey to achieve the required maximum flow velocity of 6m/s.

Optimization in design was done by using Solidworks flow simulation. The final designed wind tunnel was then simulated in ANSYS workbench to study the flow characteristics throughout the tunnel. We compared the results from solidworks and ANSYS based on pressure-velocity cut plots and graphs. From numerical analysis it is observed that a velocity of 5m/s is achieved in test section.

## REFERENCES

- [1] J.\_B. Barlow, W. H. Rae - Pope A. Low Speed Wind Tunnel Testing. 3rd Ed. Wiley-Interscience: New York, 1999. ISBN: 978-0-471-55774-6
- [2] Bell J. H., Mehta R. D. (1989) Boundary-layer predictions for small low-speed contractions. AIAA J 27(3):372-374. doi: 10.2514/3.10122.
- [3] Gultop T., "An Investigation of the effect of aspect ratio on Airfoil performance." Gazi: American Journal of Applied Sciences ISSN/EISSN: 15469239 15543641, Volume: 2, Issue: 2, Pages: 545-549, 1995.
- [4] P. Bradshaw, R. C. Pankhurst, The design of low-speed wind tunnels, Progress in Aerospace Sciences 5 (1964) 1-69
- [5] Nelton Chwee Yang Koo, "Design of Wind Tunnel (Fluid Flow Analysis)", Universiti Malasia Pahang, 2012
- [6] Nagendra Kumar Maurya, Design & Fabrication of Low-Speed Wind Tunnel and Flow Analysis G. L. Bajaj Institute of Technology & Management, INDIA
- [7] Wind Tunnel Parts." Wind Tunnel Parts. NASA, n.d. Web. Oct. 2015.
- [8] Gad-el-Hak, Mohamed. Flow Control Passive, Active, and Reactive Flow Management. Cambridge: Cambridge UP, 2000. Print.
- [9] Journal of Aerospace Technology and Management, Low Subsonic Wind Tunnel - Design and Construction.

- [10] Small-Scale Open-Circuit Wind Tunnel: Design Criteria, Construction and Calibration Mauro S. Brusca S, Lanzafame R, Famoso F, Galvagno A. and Messina M. a Department of Civil Engineering and Architecture, University of Catania, Viale A. Doria, 6, 95125, Catania, Italy.
- [11] Calautit JK, Chaudhry HN, Hughes BR, Sim LF (2014) A validated design methodology for a closed-loop subsonic wind tunnel. J Wind Eng Ind Aerodyn 125:180-194. doi: 10.1016/j.jweia.2013.12.010.
- [12] Cattafesta L, Bahr C, Mathew J (2010) Fundamentals of wind-tunnel design. Hoboken: John Wiley and Sons.
- [13] Farell C, Youssef S (1992) Experiments on turbulence management using screens and honeycombs. Project Report n<sup>o</sup> 338. Minneapolis: University of Minnesota.

## BIOGRAPHIES



**Aniruddha Patil**, Mechanical department, MIT World Peace University, Pune, India.



**Aditya Manjarekar**, Mechanical department, MIT World Peace University, Pune, India.



**Mandar Aswale**, Mechanical department, MIT World Peace University, Pune, India.



**Saeed Nakade**, Mechanical department, MIT World Peace University, Pune, India.