International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075 (Online), Volume-8 Issue-12, October 2019

Design, Fabrication and Testing of Smoke Wind Tunnel

Ankit Dangwal, Samadarshi Adhikari, Manish Kumar Bharti, Prakhar Jindal

Abstract*: Flow visualization is the direct observation of fluid flow. Moving fluids form complex patterns and understanding and visualizing those patterns is an important aspect in aerospace industry. The previously existing research and results were taken into consideration for design parameters and calculation for dimensions of various parts of the wind tunnel. Design calculations allow wind velocities in the range of 1-3 m/s at the test section. The design includes a rectangular test section of 0.06m x 0.30m cross-section for accommodation of test object of width 0.06m. The design was made in SolidWorks and was fabricated with basic materials like plywood and acrylic sheets. PC cabinet fans were used for flow circulation. Liquid wax is utilized for the purpose of smoke generation and rubber and PVC pipes were utilized for smoke containment and delivery system. Profiles that were used for testing included cylinder, sphere, cuboid, NACA airfoils and flat plate.

***Keywords*: Smoke Wind Tunnel, Smoke Generation, Flow Visualization, Flow Separation**

I. INTRODUCTION

In fluid mechanics, flow visualization provides an overall view of the flow field around a scaled model without any calculations. The flow can be made visible by employing flow visualization technique by which it would be possible to observe flow phenomena and those phenomena which are dominated by the effects of viscosity (e.g. boundary layer flows and separation) [1]. The capability to see flow patterns gives understanding onto a solution to an aerodynamic problem. An aerodynamic research tool such as wind tunnel is used to study the effects of air moving past solid objects. Wind tunnel consists of a tubular passage with the model under testing mounted in the middle also called as the test section. Air flows past the model by means of powerful fan/motor or other means. The test model equipped with sensors and instruments measures pressure distribution, aerodynamic forces and other aerodynamic-related characteristics.

By introducing dye, smoke or pigment into the flow in the wind tunnel the flow itself can be visualized. Additionally, the ability to visualize the airflow over solid objects in a wind tunnel is a valuable asset in understanding the fundamental principles of both fluid dynamics and aerodynamics.

For the system to be used in university inclusion of a non-hazardous smoke material is must and it should also have the ability to produce a uniform row of smoke sufficiently long enough to maintain its integrity
throughout the test section. The smoke wind tunnel offers the ability to visualize the important physical characteristics of fluid flows. Smoke injection is one of the means to visualize the flows. In contrast with external flows, the investigation of internal flows, either qualitatively using flow visualization or quantitatively, is not typically associated with wind tunnel measurements [2].

II. FLOW VISUALIZATION TECHNIQUES

Flow visualization can be divided into two categories: Surface flow visualization and Off-surface flow visualization. In surface flow visualization, materials such as tufts, oil, dye or clay mixture are applied onto the surface model whereas in off-surface flow visualization the use of tracers such as oil droplets, smoke particles or helium filled soap bubbles are used. All these methods require some appropriate lighting and a camera for recording the visualization. By appropriate masking of the light source if the flow field is illuminated in a plane then it is possible to examine discrete sections or slices of the flow. Optical methods such as shadowgraph and Schlieren photography can also be used to visualize compressible flows. Even computer aided visualization can be used for flow visualization.

2.1 Surface Flow Visualization

By using the chalk powder suspended in Kerosene oil the study of the flow near the surface of the square plate model can be carried out and sprayed on the model. When the flow passes the model the chalk and Kerosene mixture settles as per low lines. The direction of surface streamlines was obtained by the pattern of flow and features like the separation of flow from the surface [3].

Revised Manuscript Received on October 30, 2019.
***** Correspondence Author
Ankit Dangwal- Student*, Department of Aerospace Engineering Institute- Amity University, Haryana, Gurugram, India.
Samadarshi Adhikari- Student, Department of Aerospace Engineering , Amity University, Haryana, Gurugram, India.
Prakhar Jindal- Asst. Professor, Department of Aerospace Engineering, Amity University, Haryana, Gurugram, India.

Manish Kumar Bharti- Asst. Professor Department of Aerospace Engineering, Amity University, Haryana, Gurugram, India.

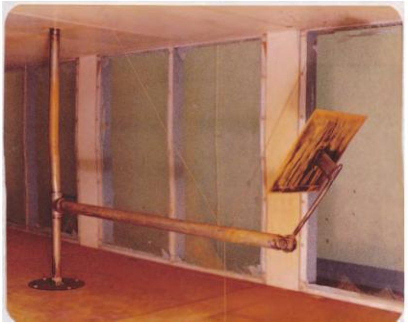

Figure 1: Surface flow visualization setup [3]

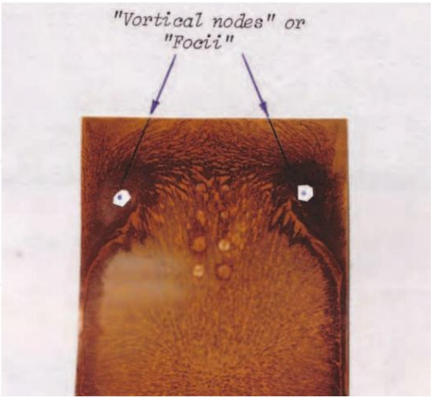

Figure 2: Surface flow visualization on wake side of flat plate [3]

The flow pattern at the surface of the model placed in the wind tunnel can be obtained quickly and easily with oil film or dots on the model surface. The mixture can be prepared from appropriate oil and fine pigment (Al_2O_3, powder, fluorescent dye, coloring pigments, graphite). This technique allows observations of the lines of separation and reattachment of the flow to the body [4].

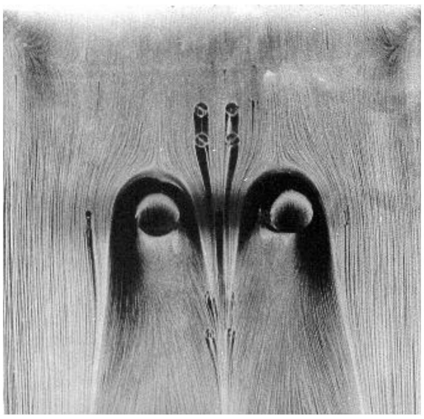

Figure 3: Flow visualization around two cylinders fixed on the plate in the large wind tunnel T-35 for $M\infty = 0.5$ with oil film [5]

Figure 4: Flow visualization around sphere for $M\infty = 0.2$ [5]

2.2 Off-Surface Flow Visualization

The visualization of the liquid flow patterns by ejection of dye is an analogy of the smoke visualization technique. The intensity of dye and water is less than that of smoke and air. Food coloring dyes, aniline, methylene, potassium permanganate, ink or fluorescent dyes can be used in water, milk and alcohol. For turbulent flow, visualization with dye is not suitable.

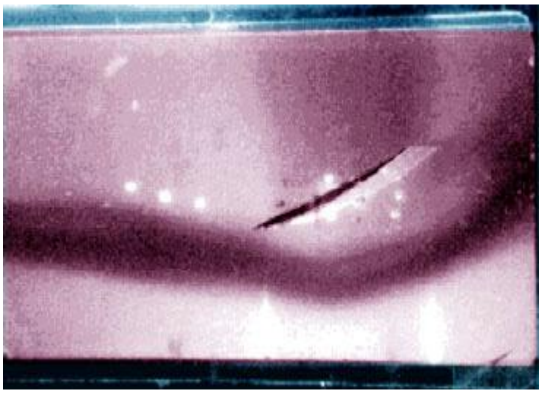

Figure 5: Flow visualization around hydrofoil in water tunnel with violet aniline dye [1]

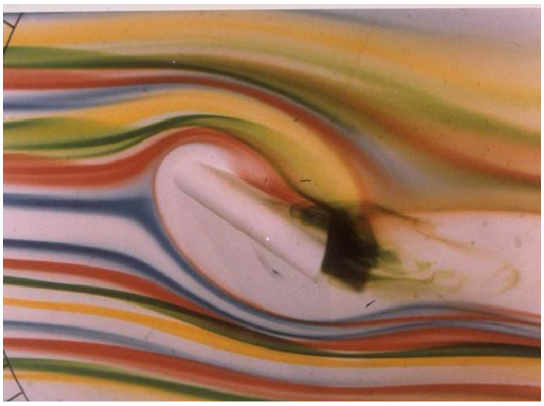

Figure 6: Water tunnel flow visualization by wall dye streaks [6]

In smoke wire technique, smoke and streams are passed through the object, as it takes the shape of the objects it reveals the type of flow and behavior of the flow. In this procedure, brushing of a thin wire is involved which forms a large number of small droplets. The oil is evaporated when the wire is heated with each droplet providing a fine streak-line in the flow.

The smoke streaks appear bright under proper illumination which can be observed and photographed. In this technique, a fine wire of about 0.1mm in diameter of Nickel Chromium steel and suitable oil is required which can vaporize quickly and a DC current to heat the wire. This technique is utilized in studying separation especially in smooth and nominal boundary turbulent flow [7].

III. SMOKE GENERATION

The use of smoke is the most convenient and inexpensive method, in which invisible air is visualized by introducing smoke into the airflow which can either be in singular streaks or multiple streams of very narrow streaks, capturing the image of the flow under generous illumination while the smoke is flowing over the body. The smoke induced in the flow follows the air currents allowing the observer to see the flow. Vaporization of liquid wax was selected as a means of producing dense smoke among several methods of smoke production. The heating element from an immersion rod is used as the heat source for vaporizing the liquid wax.
The heating element is of 300W running on 220V AC. The element is placed in steel container containing liquid wax and the whole setup is placed in a PVC pipe of diameter of 0.127m and length of 0.30cm. the PVC pipe is used for storing and containing the smoke as the liquid wax starts to evaporate. The pipe is fitted with a plastic funnel of same diameter connecting to pipe to a rubber pipe of 0.025m diameter and length of 0.91m. This other end of this rubber pipe is placed at the entrance of the plastic straw mesh which is placed at the entrance of the test chamber. The created smoke gets sucked by the fans and travels through the rubber pipe and reaches the desires destination i.e., the test section.

IV. WIND TUNNEL CONSTITUENTS AND THEIR DIMENSIONS

The dimensions of the test section are 0.50m×0.30m×0.06m which is made from 6mm and 12mm thick plywood and a 4mm thick acrylic viewing window. The second section is the contraction cone. The size of the large end of the contraction was set at 0.42m×0.18m. The smaller end of the contraction cone was set at 0.30m×0.06m to fit the test section directly and perfectly. The contraction cone made of 4mm plywood and 2mm particle board. The diffuser is 0.40m long and is glued and bolted to the test section using screws and silicon based glue. The opening of the front is of dimension 0.30m×0.06cm and the dimensions of the end opening is 38cm×12cm. A very narrow mesh made from clothing was used as flow straightener. A mesh made by aligning plastic straws is used as a flow straightener and to produce streamlines of the incoming flow containing smoke. A digital anemometer was used for velocity measurements at the inlet of contraction and test section. The velocity was measured to be in the range of 2-3m/s.

4.1 Test section dimensions

Width = 0.06m
Height = 0.30m
Length = 0.50m
Area = $0.018m^2$

4.2 Contraction Area Dimension

Length = 0.21m

$$\frac{\text{Contraction Area}}{\textit{Area of test section}} = 4 \qquad 4.1$$

Contraction Area = 4 * Cross–Sectional Area of test Section 4.2
= $0.072m^2$

Width of Contraction Cone = 3 * 0.06 4.3
= 0.18m

$$\text{Height} = \frac{\text{Contraction Area}}{\text{Width}} = 0.42\text{m} \qquad 4.4$$

4.3 Diffuser's Dimension

Length = 0.13m

$$\frac{\text{Diffuser Area}}{\text{Area of test section}} = 2.5 \qquad 4.5$$

Diffuser Area = 2.5 * Cross-sectional Area 4.6
of test section= $0.045m^2$

Width of Diffuser = 2 * 0.06 = 0.12m 4.7

$$\text{Height} = \frac{\text{Diffuser Area}}{\text{Width}} = 0.37\text{m} \qquad 4.8$$

Due to thickness of material used, 0.02m has been added to the dimension of each and every section.
Mass flow rate ($\dot{m}$) = 2 * 90cfm (empirical) = 0.0849 cubic metre/second
Assuming,
Mass flow rate ($\dot{m}$)=50 * 2 cfm = 0.0471 cubic metre/second
Cross-sectional area of the throat (A_1) = 0.03 * 0.06 = 0.0018 m^2
Cross-sectional area of test section (A_2) = 0.3 * 0.06 = 0.018 m^2
Convergence angle θ= 46.7̊
Velocity in test section V_2= 2.378 m/s
Velocity in throat V_1= 23.78 m/s

4.4 Driver system

The main aim of the fan is to maintain the flow running through the wind tunnel while compensating for pressure loss and energy dissipation. Also to get a desired flow velocity of 2-3m/s two pc cabinet fans were chosen whose specifications are as follows:

- Dimension: 120mm * 120mm * 25mm
- Voltage: 12V (DC)
- Current: 0.15 +10%A

- R.P.M.: 2000RPM ± 10%
- Air Flow: 90 CFM

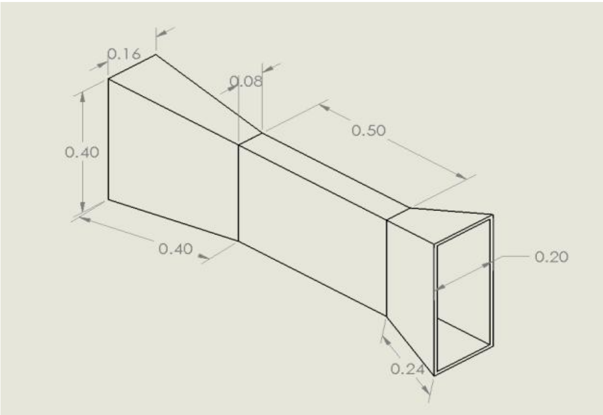

Figure 7: Schematic diagram of wind tunnel

V. RESULTS

Tests were conducted on various profiles which are; cylinder, cuboid, NACA airfoils, sphere, flat plate.

5.1 Cylinder

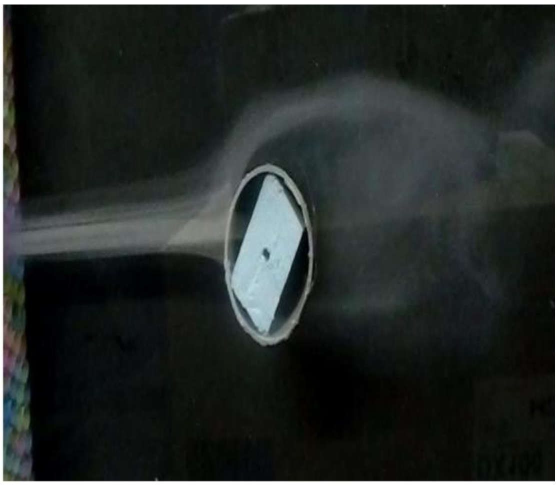

Figure 8 (a): Flow visualization over a cylindrical model

Figure 8 (b): Vortex formation above the cylindrical test object.

Above figures clearly visualizes formation of vortices and wake region at the trailing edge of the cylinder as the flow moves past the test object. The density and velocity of the flow is kept constant. The velocity gradients within the boundary layer and wake regions are much larger than those in the remainder of the flow fluid. As the flow reaches near the rear half of the cylinder, the flow now experiences an adverse pressure gradient. Consequently, the flow separates from the surface and creating a highly turbulent region behind the cylinder called the wake. The pressure inside the wake region remains low as the flow separates and a net pressure force (pressure drag) is produced.

5.2 NACA Airfoils

Figure 9 (a): Flow visualization of NACA airfoil 2412

Figure 9 (b)Flow visualization of NACA 24030 airfoil

Above figures show flow visualization over 2 airfoils, NACA 2412 and NACA 24030. Steady flow past the airfoil can be visualized. Near the angle which gives zero lift, the flow above and below the airfoil re-joins at the trailing edge except for the narrow boundary layer region where the flow is deaccelerated by friction. Since the distance between the either side of the airfoil is not same because the airfoil is cambered, the flow shows the average speed above the airfoil is greater than the flow speed beneath it.

Retrieval Number L26051081219/2019©BEIESP
DOI: 10.35940/ijitee.L2605.1081219
Journal Website: www.ijitee.org

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

Formation of small wake region at the trailing edge of NACA 2412 airfoil after which the flow becomes disturbed. Both the airfoils are kept at zero-degree angle of attack. The flow sticks to the surface till the trailing edge because of the aerodynamic shape of the airfoils.

Figure 10(a): Horse-shoe vortex formation

Figure 10(b): Vortex formation at the trailing edge of the airfoil

In above figures, NACA airfoils 2412 were chosen to be experimentally tested. Flow visualization was observed. Boundary layer separation can be visualized clearly. The airfoils were placed at zero-degree angle of attack which was then increased and decreased. When the airfoil is set at an angle which gives large lift the portion of the flow passing above the airfoil outruns the airfoil beneath. The velocity is higher near the upper surface.

5.3 Cuboid

Figure 11(a): Flow visualization over a cuboid model

Figure 11(b): Flow below the cuboid

Above figures show flow visualization over a cuboid with flow separation at the trailing edge. Flow stagnates at the leading edge of the cuboid. It shows the flow is intentionally directed below the cuboid so as to show the effect of boundary layer formation. The width of the cuboid was 0.06m. Visualization indicated that the flow possesses very complex features. The flow separates in front of the cube and at its rear corners on the top (roof) and side walls. The main vortex forms a horseshoe-type vortex around the cube and interacts with the main separation region behind the cube.

5.4 Flat Plate

Figure 12: Flow moving around the flat plate

Above figure, shows flow around a circular flat plate. Since, the flat plate has a blunt profile than an airfoil, so, when the flow strikes the surface of the flat plate the flow is dispersed all around and after the flow leaves the surface, the flow separates which ultimately results in the formation of vortices. Due to the small dimension of flat plate, the flow flows around it causing flow perturbation which can be clearly visualized.

5.5 Sphere

Figure 13(a): Flow visualization around a sphere

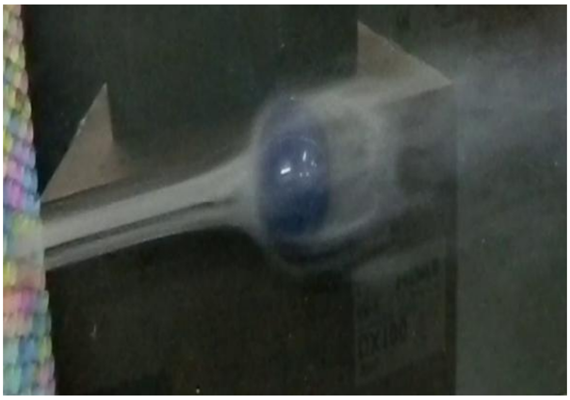

Figure 13(b): Wake and vortex formation at the trailing edge of the sphere

A sphere of diameter 0.05m was selected for analyses in the wind tunnel. It was observed that streamlines were formed and visible at flow velocities below 1m/s. Wake region was formed at the trailing edge of the model which is clearly visualized. Vortices can be observed because of the boundary layer separation. There is a pressure difference between the leading and trailing edge resulting in wake formation downstream of the spherical body. Drag is eventually created because of this pressure difference.

VI. CONCLUSION

The project's primary objective which was assessing and identifying the crucial factors and aspects which manipulate the design of a wind tunnel, some of the factors will lead to developing practical knowledge necessary when constructing fluid machinery. The surface finish or the smoothness of the interior surface of the wind tunnel, throughout, is of vital essence. Unwanted turbulence can be avoided if the surface is not irregular. The length of the diffuser should be kept significantly higher so as to reduce turbulences cause by the rotation of high speed axial flow fans. The long length keeps the flow disturbances from reaching the test section. The contraction should be wider so as to smoothen the incoming air flow. The walls of the contraction cone should somewhat resemble a cos wave, which will direct the flow more smoothly to the test section rather than a stiff joint of 15˚, which will cause vortices at the joint because of the sudden bend. All the joints and opening for the test object should be sealed properly so that no external is introduced in the main flow causing disturbances. The smoke from the liquid wax is dense and hot which should rise upwards but when it reaches the test chamber it cools down a bit and starts to settle at the bottom when the fans are turned off. The condensation of vaporized wax to liquid at room temperature is another hindrance.

VII. FUTURE SCOPE AND APPLICATION

To control the speed of a single 12V DC adapter is used which only provides a constant velocity. A DC adapter with variable voltage control can be used to regulate voltages 5-12 volts. A curve needs to be introduced in the contraction area providing a smoother airflow. a settling chamber made of fine mesh or a honeycomb structure if fitted at the entrance of the contraction cone which will break down the incoming flow and also removing or reducing the flow disturbance will allow for smoother as well as differentiable streamlines to be formed. Test chamber can be fitted with measuring instruments such a pitot static tube, or strain gauges for measurement of pressure, drag or other forces acting on the test object.

REFERENCES

1. S. Ristic, "Flow Visualization Techniques in Wind Tunnels Part I- Non Optical Methods," Scientific Technical Review, Vol. 57(1), 2007.
2. M. Mahmood, "Flow Visualization in Wind Tunnels," King Fahd University of Petroleum & Minerals, ISBN: 10.5772/21494, 2011
3. J. R. Calvert, "Experiments on the flow past an inclined disk," Journal of Fluid Mechanics, Vol. 29, 1967, pp. 691-703.
4. S. Ristic, J. Isakovic, M. Sreckovic, D. Matic, "Comparative Analysis of experimental and numerical flow visualization," Faculty of Mechanical Engineering Transacations, Vol. 34, 2006, pp. 143-149
5. S. Ristic, "Visualization of the flow, a look at the invisible," Aeromagazine, 1998, pp. 56-58
6. M. Havermann, J. Haertig, C. Rey, A. George, "Application of Particle Image Velocimetry to High-Speed Supersonic Flows in a Shock Tunnel," French-German Research Institute of Saint-Louis (ISL), Rue du Général Cassagnou F-68301, 1999.
7. S. M. Batil, T.M. Mueller, "Visualization of Transition in the flow over an aerofoil using Smoke Wire Technique," American Institute of Aeronautics and Astronautics, Vol. 19, 1981, pp.340-345.
8. M. A. G. Hernandez, A. I. M. Lopez, A. A. Jarzabek, J. M. Perales, Y. Wu and S. Xiaoxiao, "Design Methodology for a quick and Low-Cost Wind tunnel," Polytechnic University of Madrid, Spain and Beijing Institute of Technology, China, March 2013.
9. R. Ramkissoon and K. Manohar, "Design and Calibration of a Low Speed Wind Tunnel," British Journal of Applied Science & Technology, Vol. 4(20), 2014, pp. 2878-2890.