TRIBHUVAN UNIVERSITY

INSTITUTE OF ENGINEERING

PULCHOWK CAMPUS

THESIS NO:M-66-MSMDE-2020-2023

**Design and Fabrication of Smoke Flow Visualization Setup
for Existing Wind Tunnel at Pulchowk Campus**

by

Vijay Galami

A THESIS

**SUBMITTED TO THE DEPARTMENT OF MECHANICAL AND
AEROSPACE ENGINEERING
IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE
DEGREE OF MASTERS OF SCIENCE IN
MECHANICAL SYSTEMS DESIGN AND ENGINEERING**

**DEPARTMENT OF MECHANICAL AND AEROSPACE ENGINEERING
LALITPUR, NEPAL**

OCTOBER, 2023

COPYRIGHT

The author has agreed that the library, Department of Mechanical and Aerospace Engineering, Pulchowk Campus, Institute of Engineering may make this thesis freely available for inspection. Moreover, the author has agreed that permission for extensive copying of this thesis for scholarly purpose may be granted by the professor(s) who supervised the work recorded herein or, in their absence, by the Head of the Department wherein the thesis was done. It is understood that the recognition will be given to the author of this thesis and to the Department of Mechanical Engineering, Pulchowk Campus Institute of Engineering in any use of the material of this report. Copying or publication or the other use for financial gain without approval of the Department of Mechanical and Aerospace Engineering, Pulchowk Campus, Institute of Engineering and author's written permission is prohibited.

Request for permission to copy or to make any other use of the material in this report in whole or in part should be addressed to:

Head

Department of Mechanical and Aerospace Engineering

Pulchowk Campus, Institute of Engineering

Lalitpur

Nep

i

TRIBHUVAN UNIVERSITY
INSTIUTE OF ENGINEERING
PULCHOWK CAMPUS
DEPARTMENT OF AEROSPACE AND MECHANICAL ENGINEERING

The undersigned certify that they have read, and recommended to the Institute of Engineering for acceptance, a thesis entitled **"Design and Fabrication of Smoke Flow Visualization Setup for Existing Wind Tunnel at Pulchowk Campus"** submitted by Vijay Galami in partial fulfillment of the requirements for the degree of Master in Mechanical Systems Design and Engineering.

Prof. Dr. Rajendra Shrestha

Supervisor

Department of Mechanical and Aerospace Engineering

Asst. Prof. Kamal Darlami

Supervisor

Department of Mechanical and Aerospace Engineering

External Examiner, Mr. Samrat Pradhan

Deputy Manager

Civil Aviation Authority of Nepal

Committee Chairperson, Sudip Bhattrai

Head of Department

Department of Mechanical and Aerospace Engineering

Date: 6 October, 2023

Abstract

Flow visualization methods have been used to understand and demonstrate invisible flow structures in aerodynamics and other fields. Visualization in wind tunnel is a major technique used to study the flow pattern around different objects. There are mainly two methods of visualization, intrusive and non-intrusive. Non-intrusive methods using laws of optics require the variation of density in the flow which is not suitable for low-speed subsonic wind tunnels. Smoke flow visualization using smoke rake is one of the frequently used intrusive techniques for wind tunnel flow visualization. A laminar smoke filament is injected through some mechanism to the flow field in this method. The requirement for this smoke filament is to be laminar and sufficiently long to see flow around test object. In this research work, a simple smoke introducing mechanism setup was made. The smoke rake with airfoil profile of NACA 0012 with extruding pipe for smoke injection is designed and flow around it analyzed using 2d numerical simulation to see its disturbance in flow field. Its disturbance is found to be dissipated and flow to be laminar before entering test section region. For the smoke generation, fog machine of 1500 W is used and the smoke is pushed through the smoke reservoir into the smoke rake using AC fan. The test is conducted around cylinder, airfoil and ramp structure. In cylinder, turbulent wake region was clearly visible and Karman Vortex Street was also visualized. Flow separation region in cambered airfoil was also visualized. For constant, steady injection of the smoke into the wind tunnel variable speed fan is recommended in the smoke reservoir.

ACKNOWLEDGMENT

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER ONE: INTRODUCTION

1.1 Background

Flow visualization is a major subfield within experimental fluid mechanics. Making flow patterns visible have been of major interest in aerodynamics from long ago. It helps us to understand the different flow structure in many complex flow fields by enabling us to get the insights of flow from visual inspection. As most fluids are transparent in nature, we cannot visually see the underlying flow structure by human eye during a direct observation. So, the major purpose of flow visualization is to help us to understand the flow field by visually depicting the flow structure. It can be used to demonstrate the basic flow structure pattern for instructive purpose and also to understand the complex flow phenomenon, such as the turbulence. Over the time, many flow visualization techniques are developed to able us to understand the fluid motion. In many instances, flow visualization helps to develop or verify new theories of fluid flow and also validate the numerical simulation of new flow phenomenon. There are many techniques developed over the years for the flow visualization. Flow visualization can be done in numerical simulation by plotting the path-lines, contour of different flow properties. But mainly, flow visualization mainly includes the visually depicting the physical flow phenomenon in controlled experiments like in wind tunnel or in phenomenon in nature.

Flow visualization techniques also help us to get the quantitative data from the flow pictures, besides enabling us to understand the flow from visual perception. Some of these techniques, like optical techniques, help us to get the quantitative data without disturbing the flow unlike flow measurement instrument. Early pioneers of flow visualization were Osborne Reynolds, Ludwig Prandtl, and Ernst Mach. They used the flow visualization to support and illustrate their works in fluid mechanics.

In this research work, we are trying to design and fabricate a flow visualization setup for flow patterns study around different bodies by injecting smoke in the existing wind tunnel at Pulchowk Campus.

1.2 Statement of Problem

Insights through the visual perception help us to illustrate the physical processes quickly and in easy-to-understand way. So, experimentalists in fluid mechanics have devised many visualization techniques to demonstrate motion of the fluid. Among different methods of visualization, visualization of flow around the test objects in wind tunnel by introducing the smoke is one of the common methods we frequently use. The visualization of flow structure in wind tunnel helps us to depict the many basic flow structure formed around different objects. So, the current research work tries to construct a set up for flow visualization using smoke and depict basic flow structure around different objects in wind tunnel. Firstly, placing the smoke introducing mechanism inside the wind tunnel may affect the flow around the test object around which we are trying to see the air flow. And secondly, for the study of small-scale details of the basic flow phenomenon, it requires that we introduce the small and discrete smoke filaments at desired point within the flow field. So, the main problem the current research tries to solve is to make such set up while minimally affecting the flow around the test objects from introduction of the set up in the wind tunnel and studying the small-scale details of fundamentally important flow phenomenon.

1.3 Objectives of Research

Main objective
- To design and fabricate smoke flow visualization setup for the open wind tunnel

Specific objectives
1. To design smoke rake and perform numerical simulation to find out the extent of flow disturbance due to smoke rakein wind tunnel flow-field
2. To construct a smoke flow visualization setup and perform the flow visualization around different test objects

CHAPTER TWO: LITERATURE REVIEW

2.1 Wind Tunnel

Wind tunnels have been used for the study of aerodynamics around bodies from time ago. It is a primary tool for experimental study of aerodynamics. Wright brothers used simple wind tunnel in their experiments to characterize the lift and drag in their airfoil. Basically, a wind tunnel is the machine used to study the flow interaction around the bodies by holding the body stationary inside a tube and moving air through it. Different wind tunnels have been devised for small scale model testing to the full body testing and for different velocities.

There are two basic types of wind tunnels, open circuit and closed-circuit wind tunnel. Open circuit wind tunnels are used for small scale experimentation where the air flowed through the test section are expended. While in closed circuit wind tunnels, the same air is recirculated around the test section. Closed circuit wind tunnels are used where large flow of air is used as it would be inefficient economically to expend the air. According to test-section configuration wind tunnels can also be divided into open test section and closed test section.
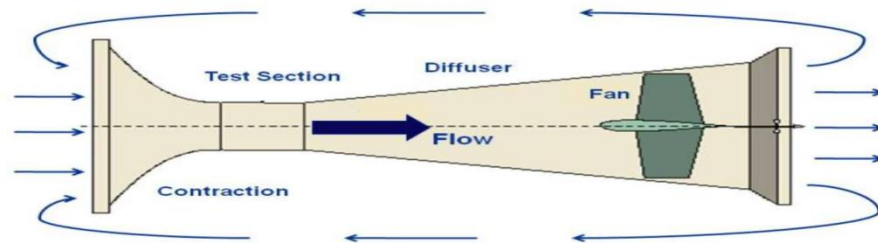
Figure 1:Schematic diagram of open circuit wind tunnel.
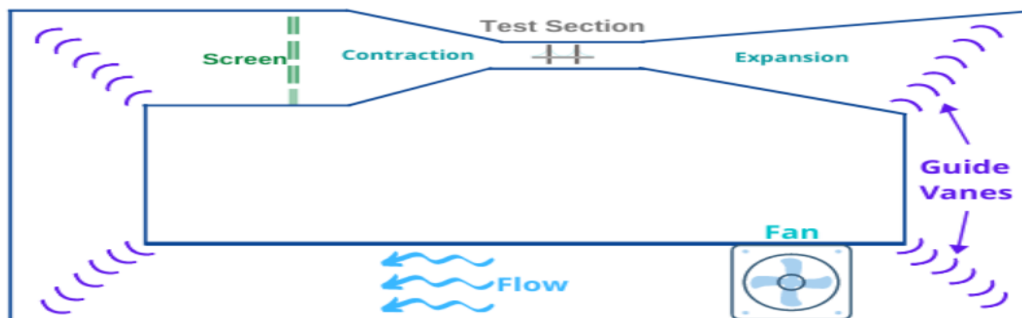
Figure 2: Schematic diagram of closed circuit wind tunnel

Low speed wind tunnel is the wind tunnel that operates around Mach number of 0.4. Smoke flow-visualization in low-speed wind tunnel is relatively easy than in high speed tunnel due to lesser Reynolds number.

Figure 3:Low speed wind tunnel at Department of Mechanical and Aersopace Engineering

The wind tunnel present in the Department of Mechanical Engineering's lab is of the open loop type. The basic construction of a wind tunnel consists of three parts: Contraction, Test Section and the Diffuser. The cross-sectional area of wind tunnel shown in the figure above is $1200mm \times 1200mm$ at inlet and $250mm \times 250mm$ at the end of contraction. The diffuser is of more than twice as long as the contractor because flow separation is highly probable during short-length expansion. For manufacturing convenience as well as for uniform expansion to prevent flow separation during diffusion, the cross section increases linearly from $250mm \times 250mm$ to $650mm \times 650mm$ cross-section over $5m$ length. There are 16 motors with centrifugal blades with rated input power $1400\ watt$ and rated output power$500\ watt$. The wind tunnel can produce flow with the velocity from$3\ m/s$ to $12\ m/s$

2.2 Flow Visualization Methods

Flow patterns around a body can be examined using experimental means of flow visualization. It provides the direct means of description of the flow around bodies without the need of data reduction and analysis. Over the years many flow

visualization techniques are developed for different applications. We can divide these techniques mainly in their groups, which are:

    i.   Surface Flow Visualization

In surface flow visualization, fluorescent dye or oil, tufts are applied to the surface of the model. And by observing the motion of these dyes or tufts over the time as flow passes around these surfaces we get the information about the boundary layer, regions of separated flows.

    ii.   Particle Tracer Methods

In particle tracer methods, we make use of tracers like smoke particles, oil droplets or helium-filled soap bubbles (Slavica, 2007). We inject this tracer particle in the flow we want to study.These particles must be visible or made visible to see their motion along with flow. If these particles are small enough to the size of fluid we are trying to visualize, the motion of these tracer particles represent the flow of the fluid itself. By using the tracer particles with density coinciding with the density of the fluid, the difference between the movement of the fluid and tracer particle can be minimized. This method is commonly used to visualize the incompressible flow as the thermodynamic state of the fluid varies in compressible flow, which is difficult to represent from the tracer particles.

    iii.   Optical Methods

The third class of the visualization uses the optical principles. Variation of fluid density causes the variation in refractive index. This allows the optical methods like shadowgraph, Schlieren photography, and interferometry to be used for visualization. Basically, these methods use the principles of optics.

2.3 Smoke Wire Technique

Smoke wire technique is one of the particle tracer methods of flow visualization. It was developed by Raspet and Moore in the early 1950's (Stephen and Thomas, 1980). This method can produce very file smoke filaments for the study of detailed structure of complex phenomenon.

The method uses the resistive heating of fine wire for the injection of smoke particles in the flow. The wire of diameter of around $0.1mm$ is used and coated with oil. The coated oil forms the small beads of oil at the surface of the wire and smoke filaments originate from these beads when wire is heated. Stainless steel or tungsten wire could be used. The wire diameter must be such that the wake region formed due to wire should not affect the small-scale details of the flow that we want to study.

The heating applied to the oil coated wire produces the smoke streak lines at the oil bead. These beads vaporize rapidly depending upon the voltage applied. For small voltage applied, continuous filaments of adequate density could be produced as long as $2s$. For even small current applied the streak lines become fainter and cannot be photographed. So, we must have arrangement of photographic technique and lighting to capture the photograph within couple of seconds. Timing circuits are used for this purpose but this complicates the whole set up for the flow visualization (Stephen and Thomas, 1980).

2.4 Smoke Flow Visualization Using Smoke Rake

Smoke flow visualization is one of the oldest methods used in wind tunnel. And it will continue to be one of the important experimental tools as a means for complex flow visualization technique. Earlier the method has been used for flow visualization in wind tunnel by introducing the large quantities of externally produced smoke either in upstream at inlet or at the inlet of the test section. In smoke flow visualization, we can use smoke, fog or vapor as a tracer particle. Its significance has increased with the better techniques developed for the injection of smokes, better lighting techniques. The application of this method is generally for the low turbulence flow in the wind tunnel.

Generally, smoke is generated outside using some kind of smoke generator and introduced in the wind tunnel. The fog generator machine can be used as a smoke source for the visualization. Thus, generated smoke is introduced through small pipes placed in front of test object or through holes on the surfaces of test object. The requirement for the smoke particle is that it must represent the flow of the air in the wind tunnel as the visual picture we get from the method is of these smoke particles. For this, the density of these particles must be same around as the fluid we are

studying. This method can be used for only the incompressible flow. In compressible flow, thermodynamic state of the flow varies as the density varies. The thermodynamic properties of the smoke are usually different from those of the fluid. This causes the difference in the motion of the smoke and the fluid particles (Wolfgang, 1987). For compressible flow, variation in density causes the variation of refractive index of the media and allows for the optical methods for flow visualization.
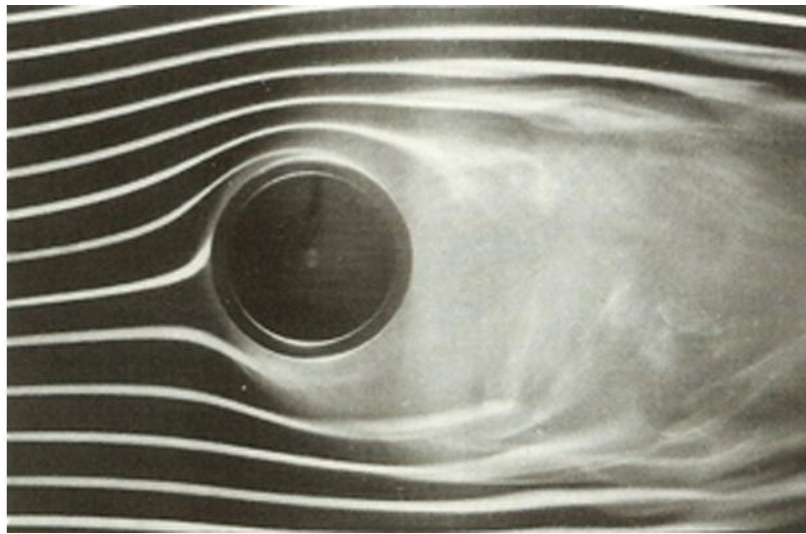
Figure 4: Smoke flow visualization around cylinder

2.5 Earlier Research

Batill and Muller, 1980, conducted an experiment on visualization of transition in the flow over airfoil using the smoke-wire technique. From their results with smoke-wire alone, if $Re_d < 20$, there was no apparent breakdown in the smoke streaklines for the entire length of test section ($1.5m$). The paper demonstrated that the smoke-wire technique provided the capability of introducing the smoke streaklines into the flowfield and, thus, provided the valuable visualization for the complex flow phenomenon.

Ravel et al, 2021, conducted research on design and analysis of smoke flow visualization apparatus for wind tunnel. They designed and constructed the smoke generator also according Preston Sweeting mist generator. And they designed

thesmoke rake profile ofNACA 0010. They installed the smoke rake at the rapid contraction section of the wind tunnel to avoid the flow diffusion and it led to streamlined flow.

Trinder and Jabbal, 2013, conducted research on "Development of a smoke visualization system for wind tunnel laboratory experiments". They set the key requirements for the systems to use a non-hazardous smoke material suitable for wind tunnel in a university setting and be able to produce a uniform row of smoke lines sufficiently long to maintain their integrity throughout the test section. For this they selected smoke rake technique with fog machine as the smoke generator. And they used water-based liquid for its non-hazardous characteristics. For the smoke rake, they used an elliptical cross-section rake with extruded pipe of $2.4mm$ diameter and $7mm$ spacing. And they conducted a successful flow visualization test around different aerodynamic and bluff body shape.

At Pulchowk Campus, Aseem et al, 2017, performed flow visualization using electronic vaporizer as smoke source. They used an electric cigarette as a smoke generator.The smoke is sucked by a piston cylinder device and the smoke stored inside the piston is blown in the air tube. The smoke from their setup lasted for around $3s$ for low speed.
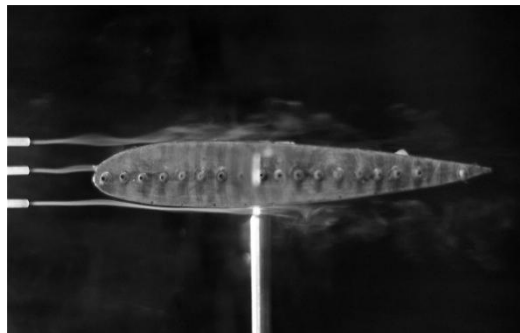
Figure 5: Smoke flow visualization (Aseem et al, 2017)

CHAPTER THREE: METHODOLOGY

This chapter describes the research methodology used for the present research work, the research design, research methods and the tools used for the research.

3.1 Research Design

The nature of the research is of numerical simulation for the first stage of the research work to find out the flow structure around the smoke rake and it is of experimental for the latter half of the research for the flow visualization setup and qualitative study of flow visualization around different test objects.

The flowchart shown in Figure 6 shows the overall methodology used for the current research work.
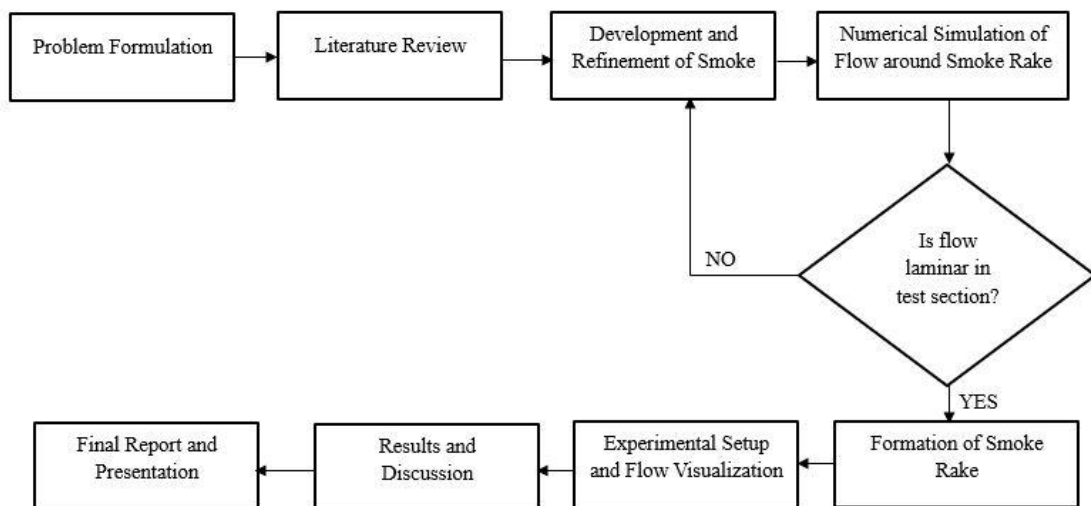
Figure 6:Flow Chart of Research Methodology

3.2 Course of Action

The entire research work is divided into different stages. Successful completion of these tasks leads to the completion of the research work. The major stages are described below.

3.2.1 Problem Formulation

Smoke flow visualization has been used for the flow patterns visualization in wind tunnel. While using this method for visualization, setting up a smoke injecting

mechanism with minimal disturbance to the flow and for visualization of the small-scale details of fundamental flow phenomenon is challenging task. Thus, in this research work we try to design and fabricate such set up and visualize the flow pattern around different test bodies.

3.2.2 Literature Review

After formulating the problem for the research, literature review is performed to find out the different techniques used by previous researchers and difficulties and problem areas for such technique's application. For the small-scale details visualization of complex flow smoke wire technique is found to be effective as it allows the injection of fine filaments of smoke at the desired location in the flow field. But the time period available for the observation is few seconds which demands the complicated set up of image capturing techniques and timing circuit for accurate capturing of the phenomenon. So, smoke flow visualization through smoke rake pipes has been selected where smoke is injected through the pipe of the carefully designed smoke rake.

3.2.3 Development of CAD model

In the next step, CAD model of the smoke rake is made in CATIA. While making a model, it requires that it should not disturb the flow-field to the extent that the flow details we are trying to visualize is affected from its introduction inside the tunnel. For this (Trindel and Jabbal, 2013) suggested an elliptical profile for the smoke rake and (Shamsuddin and Kamaruddin, 2020) used the NACA 0010 airfoils shapes for the rake cross section. For the current research, NACA 0010 airfoil is used for the rake cross section.
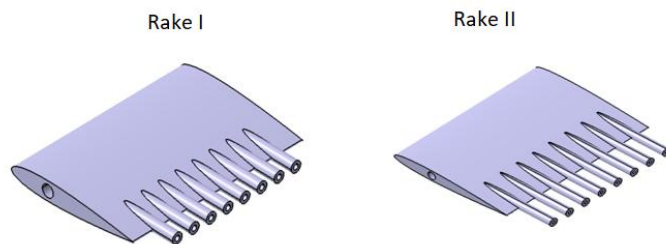
Figure 7:CAD model of smoke rake designed

3.1.4 Numerical simulation of Smoke Rake

After designing a CAD model, 2d numerical simulation of the flow around the rake is performed in ANSYS Fluent to find out the optimal operating conditions for the rake. The location for the placement of the rake and the flow velocity for which it can be used is found out from the simulation.

3.2.4 Restoration of the Wind Tunnel

The open type wind tunnel present at the Pulchowk Campus has been restored to its working condition. The replacement of three vacuum suction motor were performed and three defective switch board of the motor were also replaced. The zone of the motor replacement is shown in figure below:
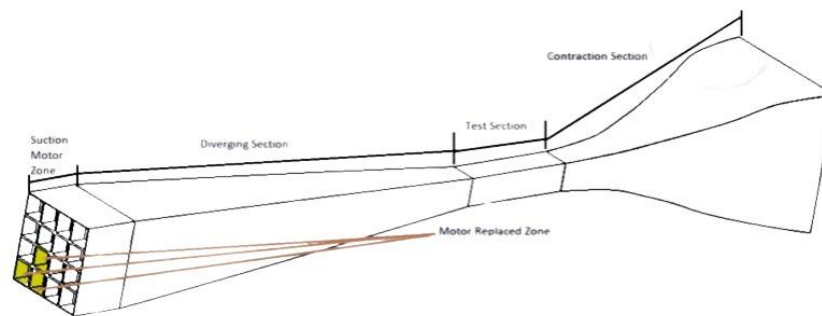
Figure 8: Schematic of wind tunnel showing replaced part

And the specification of the motor replaced is tabulated below:

Table 1:Specification of Suction Motor

No. of motors replaced	3 pieces
Power rating	1600 W
Voltage	220/240 V
Frequency	50/60 Hz

3.2.5 **Refinement and Fabrication of the smoke rake**

The next step for the research work was to refine the smoke rake model according to the simulation result and to fabricate the rake. Then, smoke rake model has been printed in resin printer present at manufacturing lab of Department ofAerospace and Mechanical Engineering, Pulchowk Campus. First lager smoke rake, Rake I is printed and flow visualization were done using it. The Rake I have the blockage area ratio of nearly 3.14%. From the Rake I, more turbulent flow of smoke was observed.Later the dimensions of the first rake were changed to obtain the second rake, Rake II with lesser blockage area than first one. Rake II has the blockage area ratio of nearly 2.69%. These values are below the recommended value in wind tunnel literature i.e. 5-10 %.
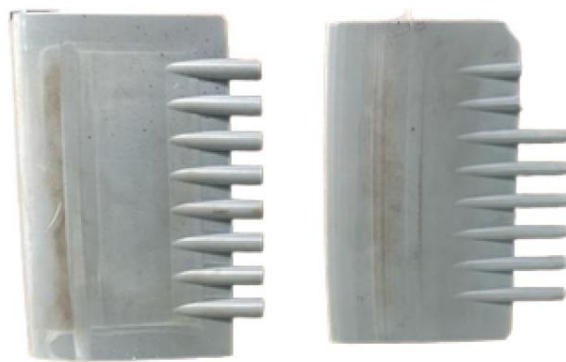
Figure 9: The 3d printed smoke-rake

The dimensions of the two-smoke rake are tabulated in Table 2 and Table 3:

Table 2: Dimensions of Rake I

Dimensions	Unit (mm)
Rake Height	140
Rake Thickness	14
Rake Width	100
Pipe Extrusion Length	60
External Diameter of Pipe	8
Internal Diameter of Pipe	4
Rake Thickness	15

Table 3: Dimensions of Rake II

Dimensions	Unit (mm)
Rake Height	140
Rake Thickness	14
Rake Width	100
Pipe Extrusion Length	60
External Diameter of Pipe	6
Internal Diameter of Pipe	2.5
Rake Thickness	12

3.2.6 **Smoke visualization set up and Experiment on different test object**

For the smoke visualization set up, firstly fog generator is obtained. The specification of the fog machine is tabulated below:

Table 4: Specification of Fog Generator

Power	1500 W
Voltage Supply	220 V
Capacity of Fog Liquid	2.5 L
Size	$530 \times 340 \times 230\ mm$
Net Weight	7 kg
Smoke Quantity	25,000 cuft/min
Warm-up time	7-8 min

The fog machine generates the smoke by pumping the fog liquid through the heat exchanger. The fog liquid gets evaporated after passing through the exchanger and the high-pressure vapor of smoke is ejected through the nozzle outlet of the fog machine. Thus, generated fog vapor mixes with the outside cool air creating a dense fog which can be used for flow visualization. The water-based glycol is used as fog liquid, which produces non-hazardous smoke.

Then, for the supply of the smoke from smoke generator to the smoke rake, a smoke reservoir is made. The reservoir was made of 20 liter water jar with pipes and valves

arrangement at its mouth for smoke outlet. The outlet from the reservoir is connected through PVC piping and plastic pipe to the inlet of the smoke rake inside the wind tunnel. A $7mm$ hole is provided on the wind tunnel wall for the insertion of smoke inlet through a pipe to the smoke rake inside the tunnel. The smoke rake also consists of $7mm$ hole in its side profile for the insertion of the pipe. This pipe is tightly fitted inside the smoke rake hole so that it can support the smoke rake inside the wind tunnel test section. For the steady flow generation to the rake two fans were installed at the bottom of the reservoir. The figure of the complete set-up is shown below:
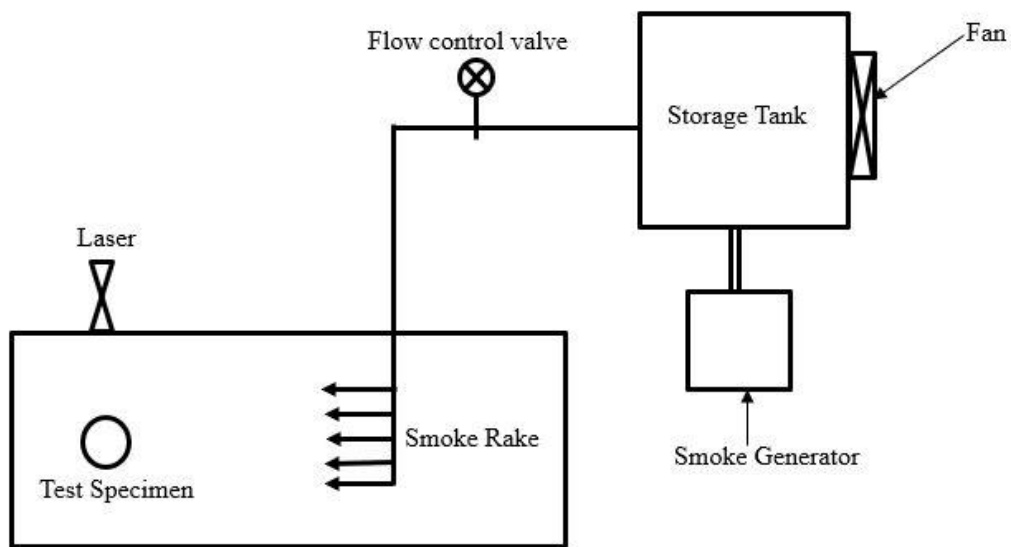

Figure 10: Schematic of Smoke Flow Visualization Setup

Figure 11: Smoke Visualization Setup

For the smoke visualization setup's operation, first, the valve of the reservoir is closed at the reservoir outlet. Then, fog machine is operated to pump the smoke inside the

14

reservoir. After enough smoke is stored in the reservoir, the valve is opened and the fan installed below the reservoir is operated to push the smoke through the piping to the smoke rake. The backside of the test section was covered using black chart paper for better visualization.

After the setup is complete, flow visualization experiment wasconducted around different test objects ranging fromcylinder, airfoil to ramp generated from oblique cut on the solid cylinder. For proper visualization of the smoke path on a frame, laser light was used.

3.2.5 Results and Discussions

The numerical simulation results for flow around the smoke rake are discussed. The flow patterns obtained around the different bodies from the experiments is analyzed and the result interpretation is done.

3.2.6 Report writing

Finally, the whole work performed is summarized and final report prepared.

3.3 Numerical Simulation Setup

Numerical simulation setup for the simulation of external flow around the smoke rake is described in this section. Numerical simulation of the flow around the designed smoke rake has been done using ANSYS Fluent. The simulation aims to find out the disturbance produced by the rake to the original flow field and helps to refine the geometry of the rake. The simulation is performed while placing the rake at the inlet zone of the test section of wind tunnel.

3.3.1 Two-dimensional simulation setup

Firstly, the geometric models of the smoke rake are created in CATIA. Then, the coordinate of the 2d profile of the smoke rake is imported in ANSYS ICEM and fluid domain with the rectangular cross section of the wind tunnel is created around it. Then the transient simulation is performed using pressure-based solver in the ANSYS Fluent.

1. Geometry

The fluid domain is the rectangular area with the 2d airfoil profile inside it. The smoke rake is placed in the middle at a distance of $150 \, mm$ from the inlet. The geometry of the 2d smoke rake profile for the simulation is shown in Figure 12. The rectangular fluid domain is of size $1588 \, mm$ by $250 \, mm$.

Figure 12: 2d smoke rake profile

The fluid domain with the smoke rake profile at the middle is shown Figure 13.

Figure 13:Fluid Domain for the Fluent simulation

2. Computational Mesh

The computational mesh is created in ANSYS ICEM mesh for the fluid domain. Structured mesh using quad cells is made up as the geometry is simple. Mesh is created using H and C type grid with C type grid around the smoke rake profile and H grid around outer wall. The total number of elements for initial simulation is 146315 quad cells. The Figure 14 shows the mesh created in the ANSYS ICEM.
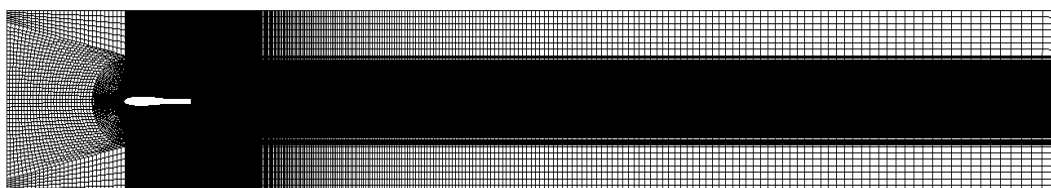

Figure 14:Mesh of the Fluid Domain Created in ANSYS ICEM

3. Solver Setup

After the completion of meshing, the CFD simulation setup is carried out. For continuity and momentum equations convergence absolute convergence criteria with value of 0.0001 is selected.

i. Material Properties

Air is used as fluid type in fluid model solver. Table 5 shows the key properties of air assigned in the setup.

Table 5:Material Properties of Air

Material	Density (kg/m^3)	Dynamic Viscosity (kg/ms)
air	1.225	1.789$10^{-5}$

ii. Model Setup

Since, the simulation is focused on the basic flow structure around the smoke rake to find out the disturbance it causes to the wind tunnel flow field, simply inviscid model is selected.

iii. Cell Zoned Condition

Fluid type (air) is assigned at the cell zone condition.

iv. Boundary Conditions

The Table 6 shows the boundary conditions assigned to the fluid model. For the inlet in the flow domain, uniform velocity inlet is taken. The assumption was that the rake will be kept at sufficient distance downstream from the contraction end so that constant speed uniform speed would have been developed. For outlet boundary condition, pressure outlet with standard atmospheric pressure at the outlet. The simulation has been performed for the two-velocity inlet condition. First case is the velocity inlet of $12 m/s$. This velocity is selected as it is the highest speed that the wind tunnel in our case is able to produce. So, the maximum disturbance the rake produces from its introduction to the flow field is expected at highest speed. For $12 m/s$ velocity inlet case the Reynolds number for

the flow taking total width of the rake as characteristic length arearound 94000. And for second case for simulation minimum velocity of $1.32 \, m/s$ for the inlet speed has been simulated. The second case has been selected as smoke visualization is expected to produce good results in low-speed flow.

Table 6:Boundary conditions of the fluid domain

Parts of the Domain	Type of Boundary Condition	Values
Inlet	Velocity-inlet	x-velocity = 12 m/s and 1.32 m/s
Outlet	Pressure Outlet	Gauge Pressure = 0 Pa
Smoke Rake	Wall	No slip
Wind Tunnel Wall	Wall	No slip

vi.   Solver Setup

Table 7and Table 8 shows the temporal and spatial discretization scheme used in Fluent.

Table 7: Temporal Discretization

Temporal Discretization	First order-implicit
Pressure-Velocity Coupling	SIMPLE
Time Step Size	0.0001 sec

Table 8: Spatial Discretization

Gradient	Least Squares Cell Based
Pressure	Second Order
Momentum	Second Order Upwind
Turbulent Kinetic Energy	First Order Upwind
Specific Dissipation Rate	First Order Upwind

3.3.2 **Three-Dimensional Simulation Setup**

Three-dimensional flow around the smoke rake was also simulated to see the flow interaction among rake pipe geometry. For three-dimensional simulation also, first, 3d rake geometry was imported in ANSYS Design Modeler and flow domain were created around it. And then mesh is generated in ANSYS with tetrahedron cells. The details of the simulation setup are described below.

1. **Domain Generation**

   The schematic of three-dimensional domain with the rake inside it is shown in Figure 15.

Figure 15 : 3d Fluid Domain

The domain has length of $1000mm$ and cross section of $250mm$ by $250\ mm$. The rake is placed at the middle of the domain and $150\ mm$ downstream from the inlet. Bodies of influences were created inside flow domain around the trailing region of the rake to capture the unsteady flow field. The Figure 16 shows the four body of influence created inside the fluid domain.
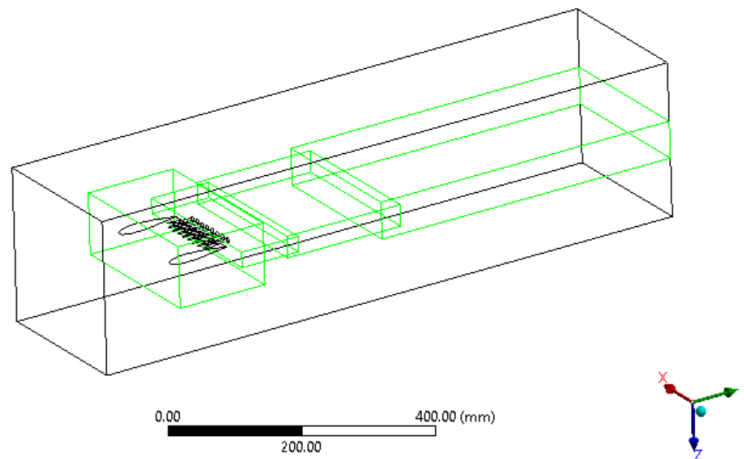

Figure 16: Fluid domain with the body of influences inside

2. Computational Mesh

The automatic mesh is generated in ANSYS with the tetrahedron cells. The 3d mesh generated is shown in Figure 17.
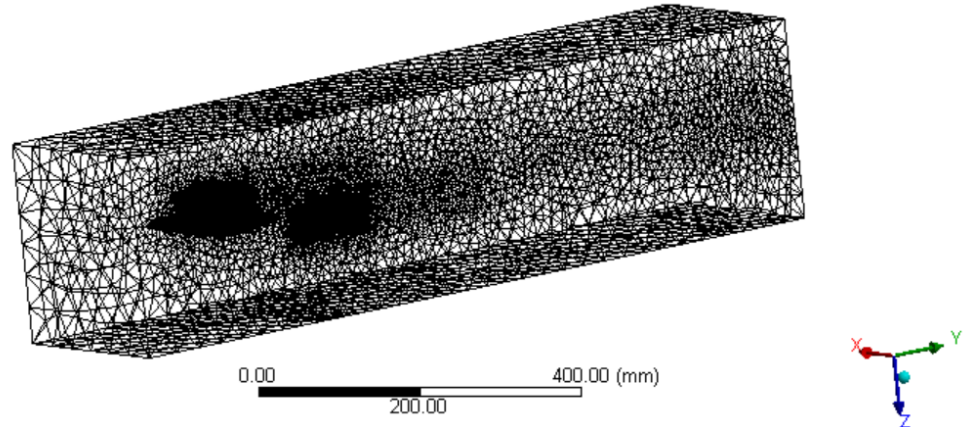

Figure 17: Three-Dimensional Mesh of Fluid Domain

3. Physics Setup

The simulation is done in ANSYS Fluent solver. The pressure-based solver is used and the solution was performed for transient simulation as formation of vortices around the pipe outlet of rake is expected. For viscous model k-omega SST model is selected. The material is same as in 2d simulation that is air. The boundary conditions for the fluid domain are tabulated in Table 9.

Table 9: Boundary conditions of the fluid domain

Parts of the Domain	Type of Boundary Condition	Values
Inlet	Velocity-inlet	x-velocity = 12 m/s
Outlet	Pressure Outlet	Gauge Pressure = 0 Pa
Smoke Rake	Wall	No slip
Wind Tunnel Wall	Wall	No slip

For pressure-velocity coupling SIMPLE scheme is used. First order implicit scheme has been used for the transient formulation. And second order implicit scheme has been used for the spatial discretization. An absolute convergence criterion of 10^{-4} has been used for continuity and momentum equation.

3.4 Cost Structure

The Table10 shows the cost for different activities for the successful completion of the research.

Table 10: Cost of the research work

S.N.	Particulars	Units	Quantity	Rate (NPR)	Total Cost (NPR)
1.	Wind Tunnel Suction Motor	-	3	1400	4200
2.	Fog Generator Machine		1	16500	16500
3.	Stationery	-	-	-	1000
4.	Documentation	-	-	-	5000
5.	Travel	-	-	-	2000
	Net Total				**28700**

CHAPTER FOUR: RESULTS AND DISCUSSION

This chapter describes the results obtained from the numerical simulation for the external flow around the smoke rake inside the wind tunnel and flow visualization experiments conducted around different test object inside the wind tunnel from the smoke visualization setup.

4.1 Two Dimensional Simulations Results

Two dimensional simulationshave been performed for two different inlet velocities of $4\ m/s$ and $12\ m/s$. The simulation was performed to see the disturbance in the wind tunnel flow field due to introduction of smoke rake.Contour of velocity and pressure is obtained and also the path lines of the flow around rake profile are created.
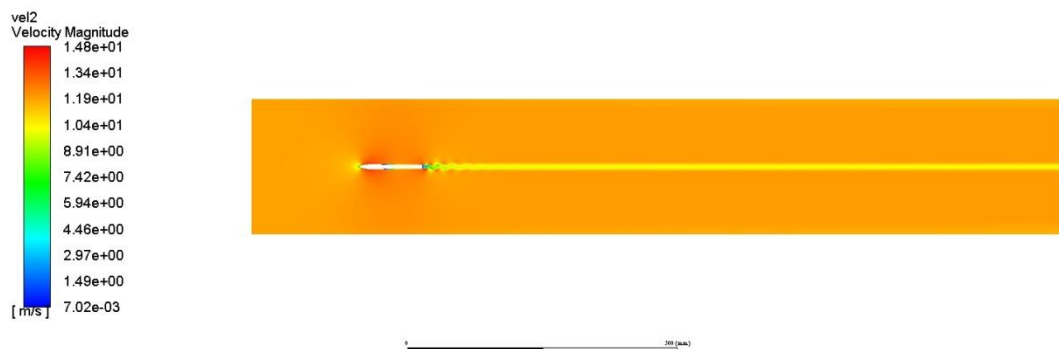
4.1.1 **Velocity Contour**

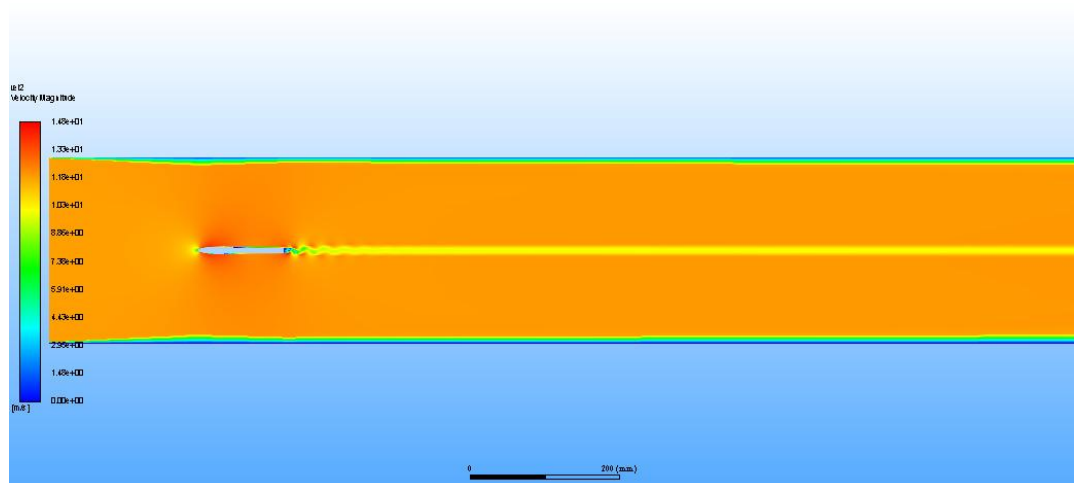Figure 18: Velocity contour at inlet velocity of $12m/s$

Figure 19:Velocity contour at $1.32\ m/s$

The velocity contour for $12\,m/s$ inlet velocity shows the maximum velocity up to $14.8\,m/s$ around the airfoil profile of the smoke rake as flow has to pass through the lower area zoned between two streamlines around curved profile. Similarly, for the $4\,m/s$ inlet velocity case velocity rise of up to $5.1\,m/s$ is obtained due to same reason. Around the wall boundary condition zone, gradual increase of velocity can be seen in the direction of interior fluid zone due to no slip boundary condition effect in these boundaries.

From the velocity contour we can observe the vortex shedding from the trailing edge of the smoke rake profile. Although the leading edge of the smoke profile has been made of symmetric airfoil profile, the pipe portion of the smoke rake at the trailing edge made for the injection of the smoke rake acts as a bluff body. The vortices are created at the back of the rake and detaches periodically from it. These vortices travel downstream with the flow and dissipate out after certain distance travel. In higher velocities, the effect of these vortices can be seen over the long distance than in lower velocity case.

4.1.2 Pressure Contour

Pressure contour of the flow shows the high-pressure zone at the leading edge of the smoke rake due to stagnation of flow in this zone. Low pressure zone is formed at the trailing edge of the rake due to formation of wake region and vortices in these areas. While most of the fluid domain has no change in pressure as the flow velocities does not change in these areas.
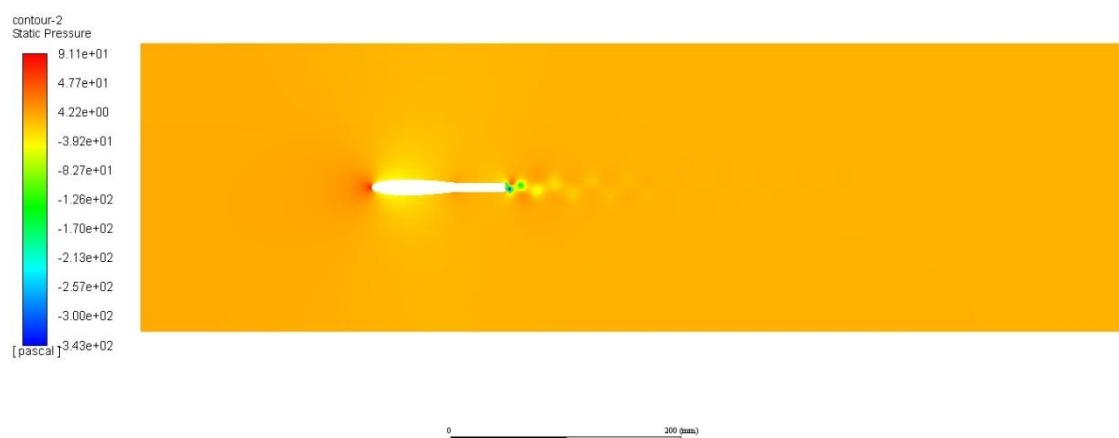

Figure 20:Pressure Contour at velocity $12m/s$

4.1.3 **Pathline**

Pathline from the inlet and the smoke rake profile is shown in the Figure 21 and Figure 22. It shows that the streamlines away from the smoke rake passes straight towards the outlet while the streamlines in the vicinity of the rake passes around the rake surface. The wavy nature of path lines is formed downstream from the trailing edge due to vortex shedding of certain frequency due to sudden change in the profile of the rake. These path lines straighten out after around $200\ mm$ downstream from the end of the smoke rake geometry. So, it will not affect the flow around the test object in our test section.
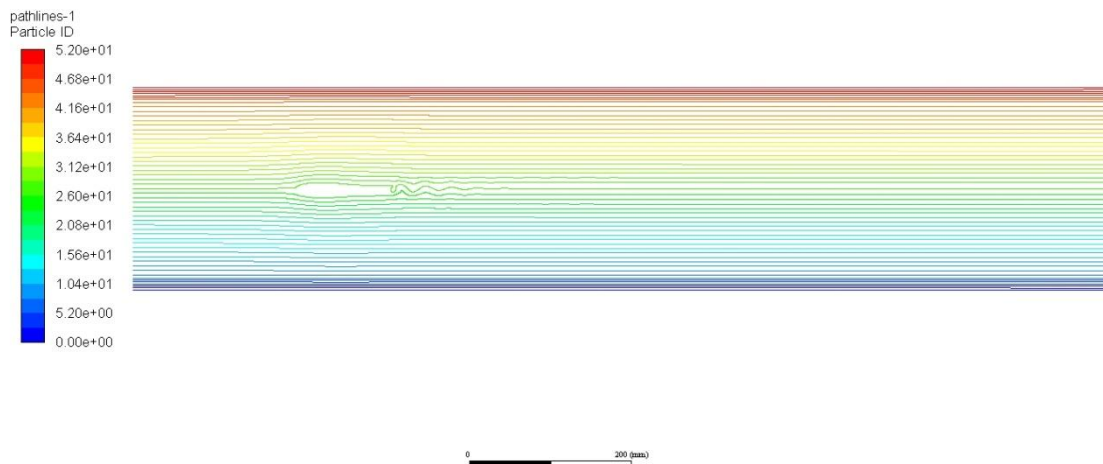

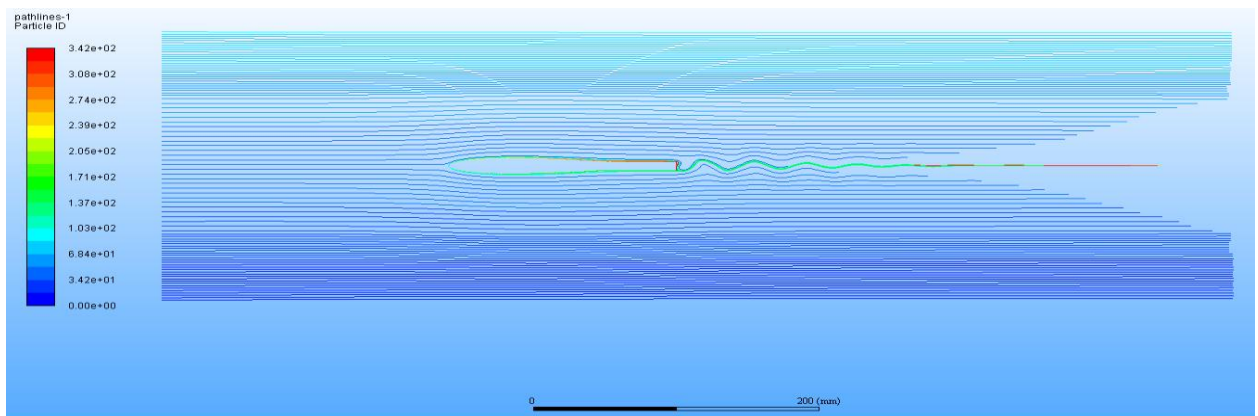Figure 21:Pathlines from inlet and smoke rake profile at $12\ m/s$

Figure 22:Pathlines from inlet and smoke rake profile at $1.32\ m/s$

4.2 Three-Dimensional Simulation Results

Three dimensional flows around the smoke rake has been simulated in ANSYS Fluent to find out if the flow vortices shedded from the rake's pipe interact and create turbulence flow downstream from the rake. Different contour plot and streamline plot were generated to see the flow structure behind the smoke rake.

4.2.1 Velocity Contour

Velocity contour of the three-dimensionsimulation is shown in Figure 23. The figure shows no velocity at the boundary wall region due to no slip boundary condition. The boundary development around the wind tunnel wall region was not captured properly but it has no effect for the flow around the smoke rake in the middle. From the flow field around the downstream region from the rake shows no interaction of flow coming from the rake's pipe geometry.
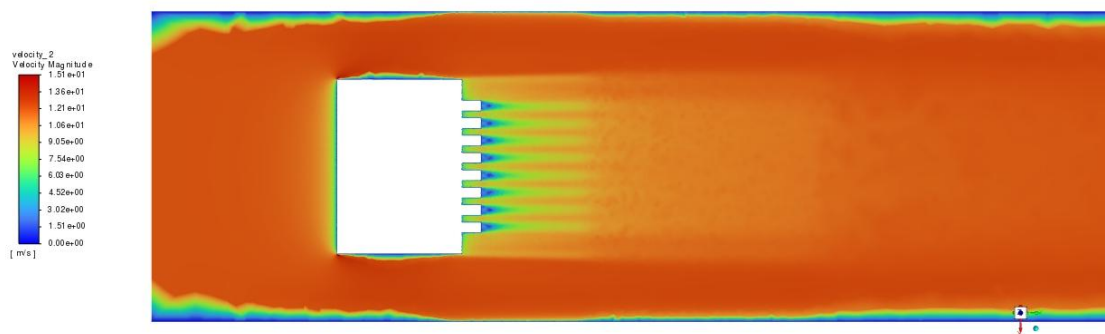
Figure 23: Velocity Contour at $12 m/s$ inlet speed

4.2.2 Pressure Contour

The pressure contour around the smoke rake flow field is shown in Figure 24. From the pressure contour we can see that the stagnation region at the leading edge of the smoke rake shows the rise in static pressure. The region at immediate downstream from the pipe of the rake shows the decrease in pressure. In this region a wake region is formed due to bluff geometry of the pipe.
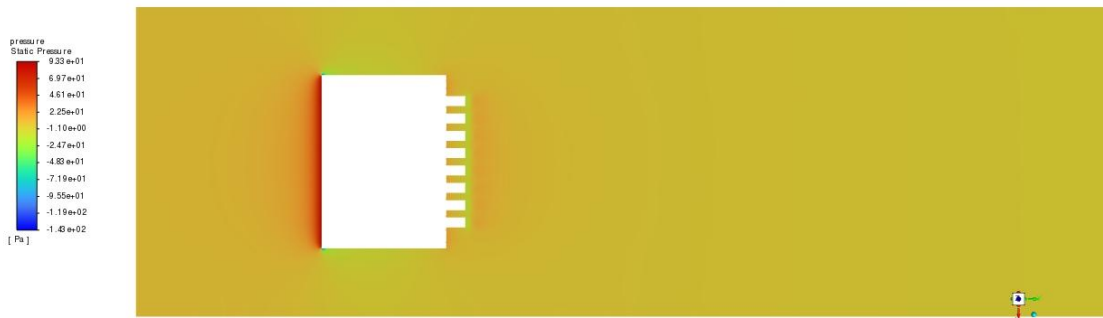
Figure 24: Pressure Contour at 12 m/s inlet speed

4.3 Smoke Flow Visualization in Wind Tunnel

4.3.1 Flow Field Downstream from Smoke Rake

To see the flow path inside the wind tunnel, smoke is introduced firstly without any test object. The test is done at various velocities and the most laminar result is obtained at velocity of 1.32 m/s which is the lowest velocity possible in our wind tunnel. The flow structure at various velocities is shown in figure below:

Figure 25:Flow Visualization at 1.32 m/s

Figure 26:Flow Visualization at 1.63 m/s
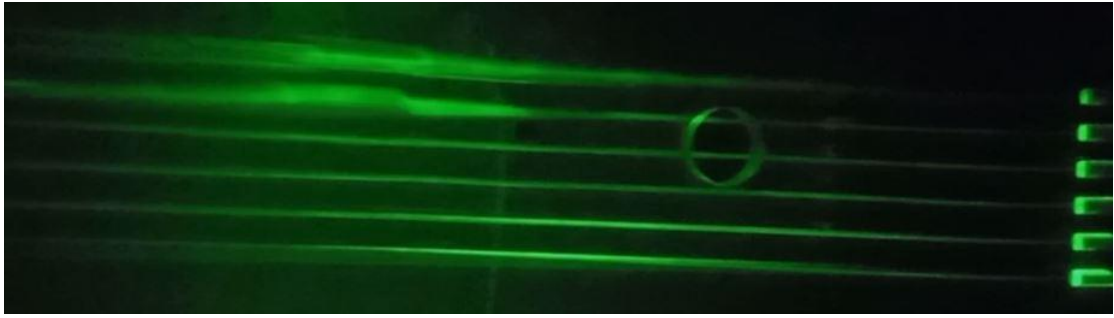
Figure 27:Flow Visualization at $1.76 \, m/s$

Figure 28:Flow Visualization at $2.61 \, m/s$

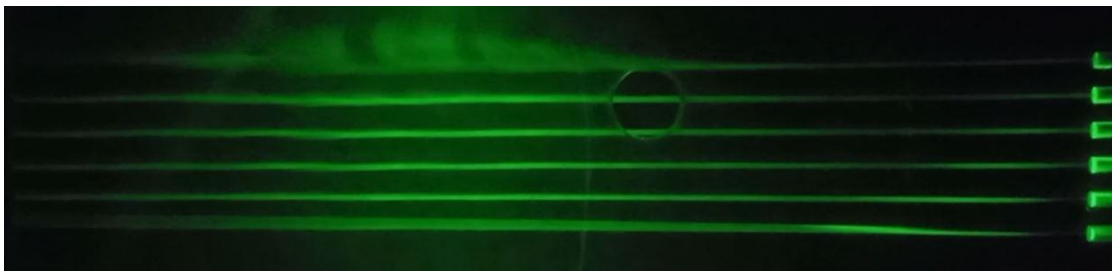From above flow field figures at various velocities, the turbulence keeps increasing as the velocity of the wind tunnel is increased. Figure 28 shows that at around $2.61 \, m/s$ the flow field are turbulent in test section region.

4.3.2 **Flow around Cylinder**

Flow around a cylindrical pipe with outer diameter of $35mm$ was performed at flow velocity of $1.32 \, m/s$. The Figure 29 show the wake region formed behind the cylinder after flow is passed around it. As cylinder is a classical bluff body flow separation can be clearly seen in Figure 29.

Figure 29: Wake region at immediate region of cylinder

Although the smoke flow's nature is turbulent, the shedding of vortices is also occurred. Reynolds number for the flow is 3,121. This is within the range of 60 to 5000 inside which vortex shedding occurs for the cylinder.
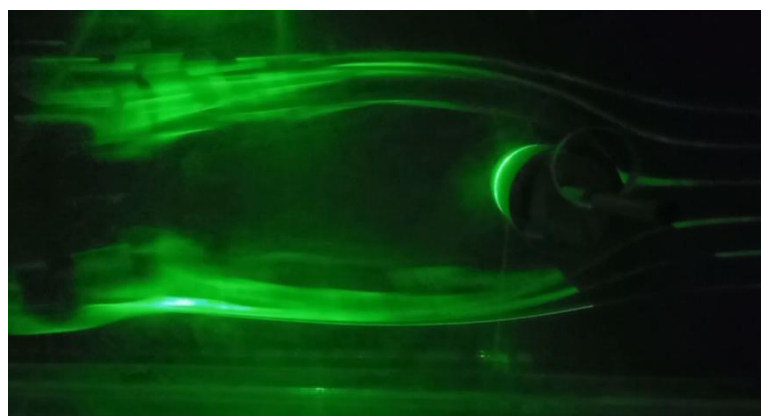
Behind the cylinder in downstream, the wake region of nearly 118 mm is formed.
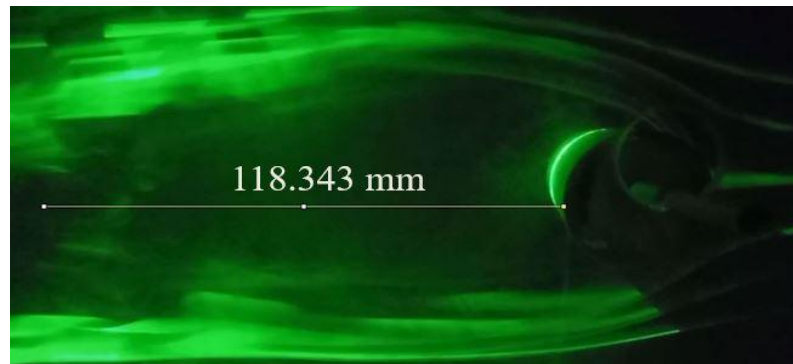
Figure 30: Wake region behind the cylinder

In Figure 31, the von karman vortex shedding behind the cylinder can be seen. The vortex shedding around the cylinder in laminar regime occurs at the Reynolds number region of $40 < Re < 150$. As we turned off the wind tunnel velocity kept decreasing and the laminar vortex shedding region was obtained. If Reynolds number could be kept in between the 40 to 150 range, either by decreasing the wind tunnel speed or by using smaller diameter cylinder, steady vortex shedding as shown in Figure 31 could be obtained.
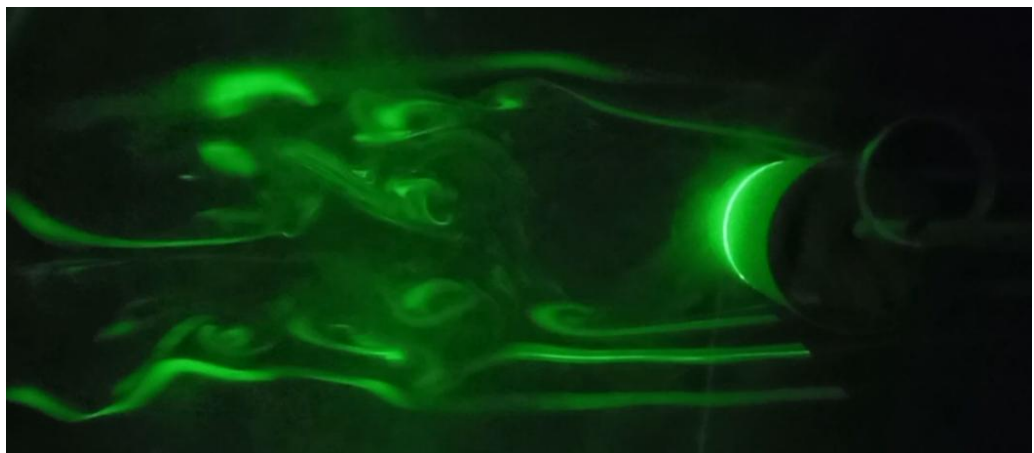
Figure 31:Vortices shed behind the cylinder

4.3.3 **Flow around Airfoil**

Flow structure around an airfoil in angle of attack about 10 degree is visualized. The airfoil has a chord length of $130mm$ and maximum thickness of $25\ mm$. The Figure 32 shows the flow around airfoil while wind is kept off. From figure we can see that the smoke filament is laminar along a longer distance. And the smoke filament has followed the airfoil profile upon encountering the airfoil surface due to Coanda effect.
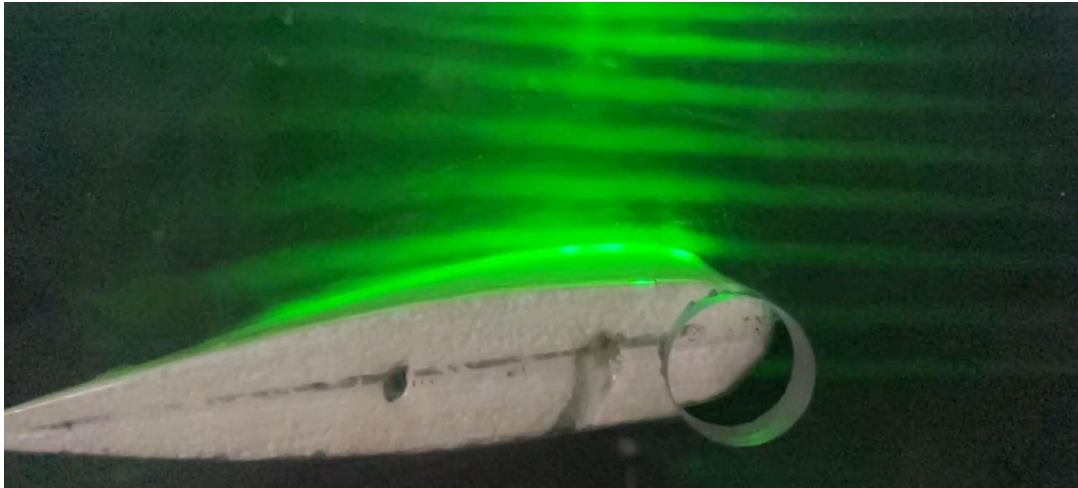
Figure 32: Flow around airfoil while wind tunnel kept off

In Figure 33, flow around the same airfoil at wind tunnel velocity of 1.32 m/s is shown. From this figure it can be seen that the smoke has become turbulent before reaching the airfoil. The smoke around the airfoil is diffused and turbulent. So, the steady flow around the airfoil could not be observed. In figure the flow can be seen separated after little distance travel downstream from the leading edge and the vortices are formed behind it.

Figure 33: Flow around an airfoil at wind tunnel speed of 1.32 m/s

In Figure 34, flow visualization around highly cambered airfoil is shown. It shows the separation of streamline at the leading-edge region.

Figure 34: Flow around a highly cambered airfoil

4.3.4 Flow around Hot Wheels Car model

Flow around the Hot Wheels car model is also visualized from the smoke visualization setup. In Figure 35 below, we can see that the flow around the car model. The smoke filament passing through the upper surface of the model is observed to follow the aerodynamic shape of the model.
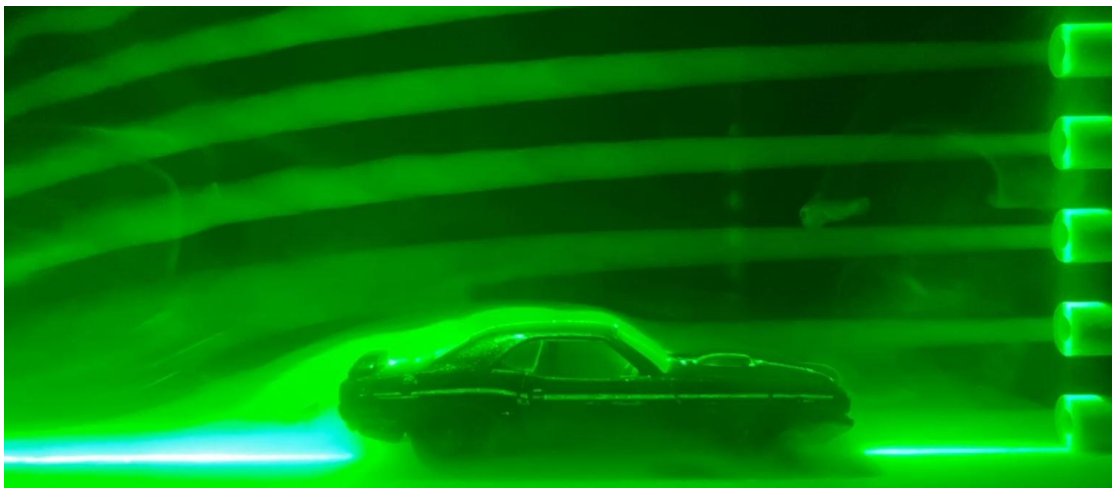
Figure 35: Flow Visualization around model car

4.3.5 Flow around Rake

A square cross sectioned rake surface of angle around 45 was created of wood and the flow past around it was visualized. In Figure 36, we can see the flow passing through the rake just before the wind tunnel was turned on. In this figure we can see the recirculating region behind the rake in downstream direction. The flow coming from

upstream was directed upward by the rake. And low-pressure region was formed behind the rake. This facilitated for the recirculation region behind the rake.
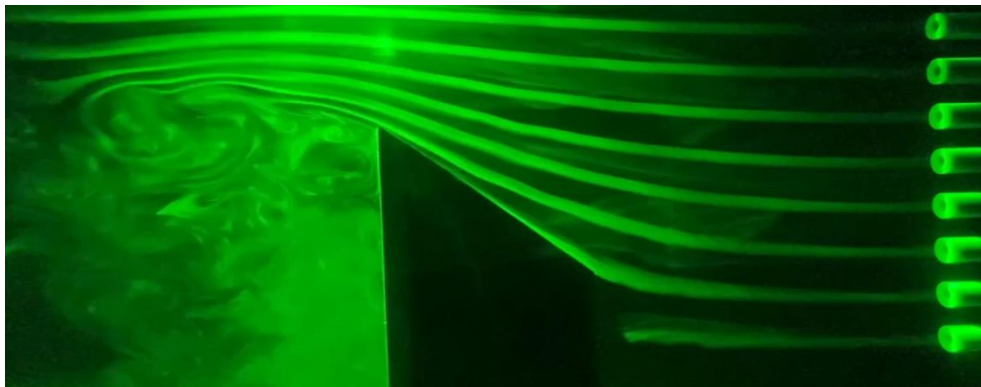
Figure 36: Flow visualization around the square rake surface

The Figure 37 shows the flow around the same rake just after the wind tunnel was turned on. From figure we can see that the laminar distinct smoke filaments are directed upward more steeply and filaments are starting to dissipate mixing with each other. After some time, all the smoke filaments mix-up and the turbulent nature of flow

Figure 37: Flow around the rake just after wind tunnel turned on

CHAPTER FIVE: CONCLUSIONSAND RECOMMENDATION

5.1 Conclusions

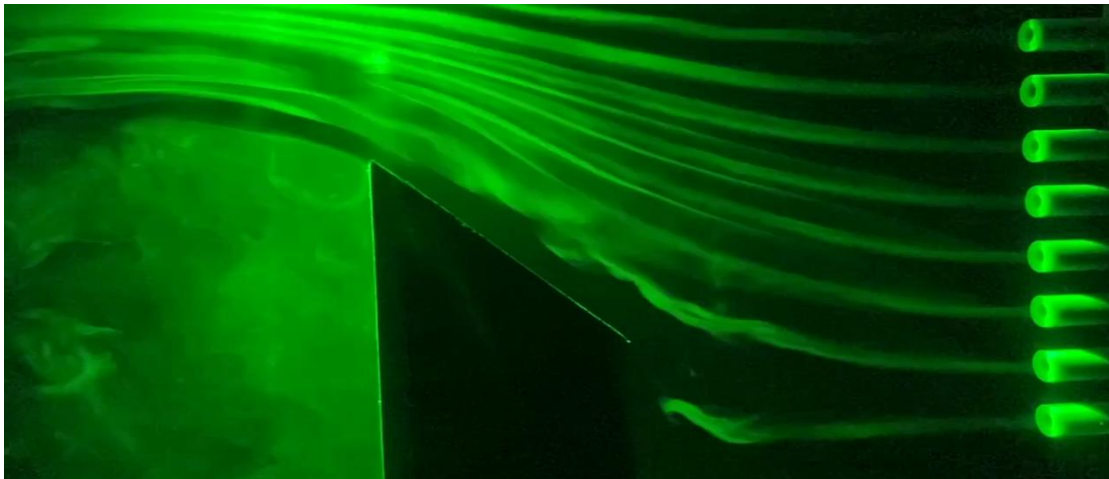An airfoil shaped smoke rake model was designed for smoke filament's introduction for flow visualization and the flow field around it in low subsonic wind tunnel flow field was solved numerically in ANSYS Fluent solver. The two-dimensional simulation around the smoke rake profile showed the generation of vortices behind trailing edge of the rake. But the pathline behind it indicated the straightening of flow streamline after around $20cm$. The three-dimensional simulation of flow field around the rake was also performed. The velocity contour for the three-dimensional flow in the plane passing through the smoke rake dividing it in two halves along the longitudinal direction indicated no turbulent characteristics.

Then, smoke visualization setup was made ready for the open type wind tunnel present at Pulchowk Campus. For the smoke generation $1500W$ fog machine was used. The smoke generated from the fog machine was stored in a reservoir and then smoke was pushed through the smoke rake from the reservoir with the help of two $12\,W$ AC fan. And flow visualization was conducted on cylinder, airfoil, Hot Wheels car model and ramp. The smoke reservoir setup and supply to the rake has still the leakage of smoke. So, there are losses of smoke and the constant steady supply of the smoke to the tunnel is not obtained. This causes the turbulence in the smoke flow. The problem could be solved after the proper sealing and the installation of small fan below reservoir for the constant supply of the smoke to the rake.

5.2 Recommendations

- Numerical simulation of flow around the smoke rake could be performed placing the rake in contraction section to obtain more streamlined flow-field.
- Further numerical study could be performed on flow structure from the smoke rake pipe spacing to obtain the optimal spacing to generate the fine smoke filaments.
- The smoke outlet velocity at the smoke rake could be controlled using variable speed fan with higher flow rate.

REFERENCES

1. Lerner, J. C., & Boldes, U. (Eds.). (2011). *Wind tunnels and experimental fluid dynamics research*. BoD–Books on Demand.

2. Ristić, S. (2007). Flow visualisation techniques in wind tunnels part i–non optical methods. *Scientific Technical Review*, *57*(1), 39-50.

3. Raval, D., Jain, S. V., Acharii, A. M., & Ghosh, K. (2021, November). Design and analysis of smoke flow visualization apparatus for wind tunnel. In *IOP Conference Series: Materials Science and Engineering* (Vol. 1206, No. 1, p. 012014). IOP Publishing.

4. Shamsuddin, M. S. M., & Kamaruddin, N. M. (2020, September). Design and Development of a Small-Scale Wind Tunnel for Flow Visualization. In *IOP Conference Series: Materials Science and Engineering* (Vol. 920, No. 1, p. 012001). IOP Publishing.

5. Batill, S. M., & Mueller, T. J. (1981). Visualization of transition in the flow over an airfoil using the smoke-wire technique. *AIAA journal*, *19*(3), 340-345.

6. ANSYS, Inc. (2016) ANSYS Fluent User's Guide, Release 17.2.

7. Barlow, J. B., Rae, W. H., & Pope, A. (1999). *Low-speed wind tunnel testing*. John wiley & sons.

8. ANSYS ICEM CFD.(2012) User Manual.

9. Trinder, M., & Jabbal, M. (2013). Development of a smoke visualisation system for wind tunnel laboratory experiments. *International Journal of Mechanical Engineering Education*, *41*(1), 27-43.

10. Settles, G. S. (1986). Modern developments in flow visualization. *AIAA journal*, *24*(8), 1313-1323.

11. Ristić, S. (2007). Flow visualization techniques in wind tunnels–optical methods (part ii). *Scientific Technical Review*, *57*(2), 38-49.

12. Merzkirch, W. (2007). Flow Visualization. In: Tropea, C., Yarin, A.L., Foss, J.F. (eds) Springer Handbook of Experimental Fluid Mechanics. Springer Handbooks. Springer, Berlin, Heidelberg.

Appendix A: Experimental Setup

The major components of the experimental setup are:

1. Fog machine
2. Smoke Reservoir
3. Smoke circulating fan
4. Smoke rake
5. Laser light source

(a) Fog-Machine

(b) Smoke Reservoir

(c) Fan Arrangement

(d) Smoke-Rake

(e) Fog Machine with Reservoir

(f) Laser light source

(g) Flow Visualization Setup

Appendix B: Drawing of Smoke-Rake

R4.5

8
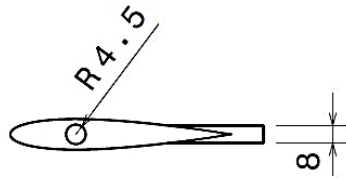
Bottom view
Scale: 1:3
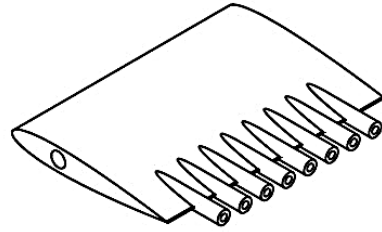
Isometric view
Scale: 1:3

116

100
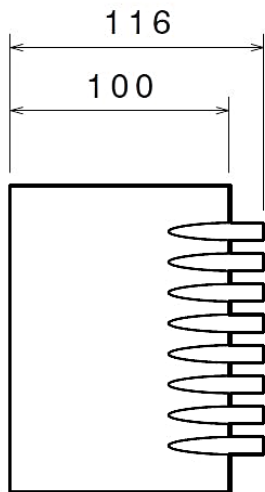
Front view
Scale: 1:3

140
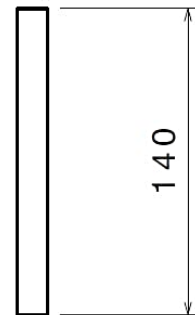
Left view
Scale: 1:3

DESIGN AND FABRICATION OF SMOKE FLOW VISUALIZATION SETUP FOR OPEN TYPE WIND TUNNEL AT PULCHOW CAMPUS			
PART DRAWING OF SMOKE RAKE I			
SCALE	1:3	DRG. SHEET NO.	1 OF 2
PULCHOWK CAMPUS, IOE, TU			
Dimensions are in mm.			

Smoke Flow Visualization

ORIGINALITY REPORT

16%

SIMILARITY INDEX

PRIMARY SOURCES

1. elibrary.tucl.edu.np
   Internet
   528 words — 6%

2. www.researchgate.net
   Internet
   85 words — 1%

3. D Raval, S V Jain, A M Acharii, K Ghosh. "Design and analysis of smoke flow visualization apparatus for wind tunnel", IOP Conference Series: Materials Science and Engineering, 2021
   Crossref
   79 words — 1%

4. WOLFGANG MERZKIRCH. "Introduction", Elsevier BV, 1987
   Crossref
   64 words — 1%

5. S. BATILL, T. MUELLER. "Visualization of the laminar-turbulent transition in the flow over an airfoil using the smoke-wire technique", 11th Aerodynamic Testing Conference, 1980
   Crossref
   49 words — 1%

6. repository.ju.edu.et
   Internet
   30 words — < 1%

7. www.mr-cfd.com
   Internet
   26 words — < 1%

8 Dangre, Asmita. "A Clinical Study of Extra-Cranial Complications of Chronic Suppurative Otitis Media", Rajiv Gandhi University of Health Sciences (India), 2023
ProQuest

25 words – < 1%

9 Flow Visualization VI, 1992.
Crossref

25 words – < 1%

10 www.vti.mod.gov.rs
Internet

22 words – < 1%

11 Steven A. Brandt, Randall J. Stiles, John J. Bertin, Ray Whitford. "Introduction to Aeronautics: A Design Perspective, Third Edition", American Institute of Aeronautics and Astronautics (AIAA), 2015
Crossref

20 words – < 1%

12 publications.lib.chalmers.se
Internet

20 words – < 1%

13 "Technology Innovation in Mechanical Engineering", Springer Science and Business Media LLC, 2022
Crossref

19 words – < 1%

14 aviation.sosu.edu
Internet

18 words – < 1%

15 repository.tukenya.ac.ke
Internet

17 words – < 1%

16 www.scielo.br
Internet

15 words – < 1%

17 ethesis.nitrkl.ac.in
Internet

14 words – < 1%

18 uknowledge.uky.edu
    Internet
                                    13 words — < 1%

19 iopscience.iop.org
    Internet
                                    11 words — < 1%

20 www.coursehero.com
    Internet
                                    11 words — < 1%

21 utpedia.utp.edu.my
    Internet
                                    10 words — < 1%

22 www.rizshopping.com.pk
    Internet
                                    10 words — < 1%

23 G. J. Hwang, M. H. Lin. "Estimation of the Onset of   9 words — < 1%
    Longitudinal Vortices in a Laminar Boundary Layer
    Heated From Below", Journal of Heat Transfer, 1995
    Crossref

24 Krishna Thakkar, Akanksha Kesarwani, Karar   9 words — < 1%
    Ahmad Khan, Rahul Sunil, Vinayak Malhotra, B. T.
    Kannan. "Experimental Investigation on Laser Visualization of
    Flow Vortices", 2019 IEEE Aerospace Conference, 2019
    Crossref

25 Thomason, Nicole M.. "Experimental Investigation   9 words — < 1%
    of Suction Slot Geometry on a Goldschmied
    Propulsor.", California Polytechnic State University, 2023
    ProQuest

26 Hossam Hamid, Rafea Mohamed Abd El Maksoud.   8 words — < 1%
    "A comparative examination of the aerodynamic
    performance of various seashell-shaped wind turbines",
    Heliyon, 2023
    Crossref

27 Nathalia Cavichiolli de Oliveira. "Gut microbiota of the rice and corn strains of >i<Spodoptera frugiperda>/i<: diversity and function", Universidade de Sao Paulo, Agencia USP de Gestao da Informacao Academica (AGUIA), 2021
Crossref Posted Content

8 words — < 1%

28 Nqombolo, Azile. "Design of Metal Organic Framework Anchored on Graphene Oxide and Their Incorporation into Polymer Membranes for Water Treatment", University of Johannesburg (South Africa), 2023
ProQuest

8 words — < 1%

29 Pengfei Zhang, Jiajun Lu, Kangbo Yang, Haipeng Chen, Yuqi Huang. "A 3D simulation model of thermal runaway in Li-ion batteries coupled particles ejection and jet flow", Journal of Power Sources, 2023
Crossref

8 words — < 1%

30 Sejong Chun. "Chapter 9 Air Speed Measurement Standards Using Wind Tunnels", IntechOpen, 2011
Crossref

8 words — < 1%

31 Sridhar Ravi, P. Petersen, S. Watkins, M. Marino, J. Watmuff. "AERODYNAMIC PERFORMANCE AND FLOW STRUCTURE OVER A THIN AIRFOIL UNDER SMOOTH AND TURBULENT CONDITIONS AT LOW REYNOLDS NUMBERS", Journal of Flow Visualization and Image Processing, 2011
Crossref

8 words — < 1%

32 Subrahmanya Veluri, Anwar Ahmed, Christopher Roy, Rifki Rifki. "Preliminary RANS Simulations and Experimental Study of a Simplified Tractor/Trailer Geometry", 24th AIAA Applied Aerodynamics Conference, 2006
Crossref

8 words — < 1%

33 archive.org
Internet   8 words — < 1%

34 digibuo.uniovi.es
Internet   8 words — < 1%

35 dr.ntu.edu.sg
Internet   8 words — < 1%

36 eureca18.taylors.edu.my
Internet   8 words — < 1%

37 getd.libs.uga.edu
Internet   8 words — < 1%

38 purehost.bath.ac.uk
Internet   8 words — < 1%

39 www.mdpi.com
Internet   8 words — < 1%

40 Eichermueller, Michael Christian. "A Performance
Study and Characterization of a Single use   7 words — < 1%
Pharmaceutical Vibrational Mixer Using Computational Fluid
Dynamics.", California Polytechnic State University, 2023
ProQuest

41 Huan J. Keh, Tse Y. Huang. "Diffusiophoresis of
colloidal spheroids in symmetric electrolytes",   7 words — < 1%
Colloids and Surfaces A: Physicochemical and Engineering
Aspects, 1994
Crossref

42 Zhang, Shanying. "Optical Diagnostics Applied to
Natural and Aerodynamically Excited Turbulent   7 words — < 1%
Jets.", The University of Manchester (United Kingdom), 2003
ProQuest

43 dissertations.umi.com
Internet   7 words – < 1%

44 Martin Trinder, Mark Jabbal. "Development of a   6 words – < 1%
    Smoke Visualisation System for Wind Tunnel
    Laboratory Experiments", International Journal of Mechanical
    Engineering Education, 2013
    Crossref

45 Talu, O.. "Measurement and analysis of   6 words – < 1%
    oxygen/nitrogen/ 5A-zeolite adsorption equilibria
    for air separation", Gas Separation and Purification, 1996
    Crossref

46 ro.uow.edu.au   6 words – < 1
    Internet