Fluid Mechanics Lab
ME-224

DEPARTMENT OF MECHANICAL ENGINEERING
IIT BOMBAY

Spring 2022

NAME:

ROLL NO.:

BATCH:

Expt. No.	E 1	E 2	E 3(a)	E 3(b)	E 4	E 5
Marks Obt.						
Max. Marks	10	10	5	5	10	10

Expt. No.	E 6	E 7	E 8	E 9	E 10(a)	E 10(b)
Marks Obt.						
Max. Marks	10	10	10	10	5	5

Table of Contents

Important Instructions

1. This laboratory course will be conducted in the online mode in the first half of spring 2022 semester. There is no time slot allotted to it. If is decided to change the mode post-midsem you will be communicated accordingly. The lab assignments for each experiment will be uploaded on Moodle at the beginning of each week. **The assignment needs to be completed and submitted before 11:59 pm of on the Sunday of that week**.
2. For each week's lab assignment students should go through the description of the experiment corresponding to the assignment. They can refer to the related theory from other sources such as text books if required.
3. Links to the videos of the experiment corresponding to that particular week will be provided to the students on Moodle along with the experimental readings.
4. Students should do the required calculations, record the results, observations and conclusions in the manual, draw the required graphs and upload a scanned copy of their work on Moodle before the mentioned deadline for the respective lab assignments.
5. Conclusions are to be well thought about and they should be such that the theory taught in the course should be connected with the experimental results.
6. Mid-sem and end-sem exams would be a closed book written examinations. **Weightages are 30% for lab assignments, 35% for mid-sem exam and 35% for end-sem exam**.

Safety Instructions: Fluid Mechanics Lab

Although the course will be conducted in the online mode, following instructions are important to remember for anytime when students enter the Fluid Mechanics lab

1. Closed toed shoes are COMPULSORY. If you are not wearing them, please go to your hostel, wear shoes and come back.
2. If you have long hair, please tie it for the duration of the lab.
3. Avoid wearing loose clothing to the lab. If wearing a full hand shirt, either button the cuffs or fold the sleeve.

4. Stay at a safe distance (approximately 1 ft away) from all the moving parts in the setup. Readings should be taken from a distance.
5. Always be wary of moving parts in the pump and motors. Immediately switch them off if any moving part breaks.
6. Water in the tanks may be corrosive, so avoid contact with water as much as possible. Wash your hands carefully after the experiment.
7. Switch off your mobile phones for the duration of the laboratory.
8. Watch out for cables, wires, etc. lying on the floor to avoid slips, trips & falls.

Schedule of Experiments

Week-wise schedule of experiments will be provided on Moodle

List of Instructors

Faculty:

Prof. Ankit Jain
Email: a_jain@iitb.ac

Prof. Avishek Ranjan
Email: avishekr@iitb.ac.in

Prof. Janani Srree Murallidharan
Email: js.murallidharan@iitb.ac.in

Prof. Neeraj Kumbhakarna
Email: neeraj_k@iitb.ac.in

Teaching & Research Assistants:

Gajendra Verma
Email: gkviitb@gmail.com

Akshay Kumar
Email: 174106005@iitb.ac.in

Anandkumar Patel
Email: 193109005@iitb.ac.in

Arunjoy Baruah
Email: 194100015@iitb.ac.in

John Pinto
Email: 184106003@iitb.ac.in

Maitreya Sinha
Email: maitreya_s@iitb.ac.in

Pratik Bagal
Email: 203100003@iitb.ac.in

Ankush Pujari
Email: 194100006@iitb.ac.in

Ravi Kant
Email: 204100019@iitb.ac.in

E 1: CALIBRATION OF VENTURI METER AND ORIFICE PLATE

AIM: To calibrate Venturimeter and Orifice plate for a given fluid and study the variation of coefficient of discharge C_d with Reynolds Number.

APPRATUS: Venturi meter, mercury manometer, stop watch, calibrated measuring tank, orifice plate.

THEORY: Venturimeter is used to measure the rate of flow through a pipe. Venturimeter consists of a converging portion, throat and a diverging portion. The function of the converging portion is to increase the velocity of the fluid and temporarily lower its static pressure. The pressure difference between inlet and throat is developed. This pressure difference is correlated to the rate of flow. The expression for theoretical flow rate is obtained by applying the continuity equation and Bernoulli's equation at inlet and throat section, and assuming the fluid to be ideal is given by:

Theoretical mass flow rate: $Q = A_2 \sqrt{\frac{2\,\Delta p}{\rho\,(1-\beta^4)}}$

Same equation for theoretical discharge holds good for orificemeter also. Construction of orificemeter is simplest amongst all the flow meters in that it consists of a plate with a hole drilled in it. In principle, it is essentially similar to a venturi since it obstructs the flow of fluid.

However, due to the absence of guiding passage on the downstream passage of the orificemeter, fluid comes out in the form of a free jet. This difference in the flow physics of the two flow meters leads to difference in the value of discharge coefficient and irrecoverable pressure loss even when the area ratios for two are identical.

CALIBRATION OF FLOWMETERS:

All the flow-meters need calibration a prior where a known quantity of fluid is passed through the flow-meter and the differential pressure across the flow meter is related to the actual discharge through a discharge coefficient given as the ratio of actual to theoretical mass flow rate.

PROCEDURE:

1) Check if all the valves are closed and then start the motor.
2) Now open the bypass valve so that all water is discharged into sump tank.
3) Now open the outlet valve of venturi meter keeping the outlet valve of orifice meter closed.
4) First open the air valves of manometer and then open the venturi meter valves of manometer. Remove all the air bubbles through circuit.
5) Adjust the discharge by closing the bypass valve. Note down the manometer reading. Calculate the theoretical discharge Q_{th}..
6) Note down the time for collection of 10 litres of water in the measuring tank and determine the actual discharge Q_{act}. Calculate the coefficient of discharge C_d. Repeat the procedure for five mass flow rates for both Venturi meter and orifice meter.

$$Re_D = \frac{4\,\rho\,Q_{act}}{\pi\,D\,\mu}$$

7) ISO 5167 specifies value of discharge coefficient for orifice meter as a function of diameter ratio β and Reynolds number Re_{D1}-

$$C_d = 0.5961 + 0.0261\beta^2 - 0.216\beta^8 + 0.000521\left(\frac{10^6\beta}{Re_D}\right)^{0.7}$$

8) Calculate the value of discharge coefficient for the orifice meter according to ISO standard. Also note that the value of discharge coefficient for Venturi meter as specified by ISO 5167 is 0.984.

GRAPHS: Plot a graph of C_d vs Re_D obtained by volume flow rate measurement and ISO 5167 on a single graph paper.

RESULTS/DISCUSSIONS:

SOURCES OF ERROR:

CALCULATIONS:

1. Theoretical mass flow rate,

$$Q_{th} = A_2 \sqrt{\frac{2\,\Delta p}{\rho_w (1 - \beta^4)}}$$

$$\beta = \frac{d}{D}$$

D = inlet diameter, d = throat diameter
A_1 and A_2 are areas at inlet and throat

2. Conversion of head in mercury column to water column,

$$h_w = \left(\frac{\rho_{Hg}}{\rho_w} - 1\right) h_{Hg}$$

$$\Delta p = \rho_w \, g \, h_w$$

3. Coefficient of discharge (experimental): $C_d = \frac{Q_{act}}{Q_{th}}$

4. Coefficient of discharge (ISO),

$$C_d = 0.984$$ (Venturimeter)

$$C_d = 0.5961 + 0.0261\beta^2 - 0.216\beta^8 + 0.000521\left(\frac{10^6 \beta}{Re_D}\right)^{0.7}$$ (Orificemeter)

Useful Data:

Density of mercury = 13600 kg/m^3
Density of water = 1000 kg/m^3
Viscosity of water, μ = 0.0007975 Pa-s

OBSERVATION TABLE:

Venturimeter:

Supply pipe diameter = 21mm

Venturimeter inlet (D) and throat (d) diameter = 21.5mm and 15mm

Sr. No.	Volume, *V* (litres)	Time, *t* (sec)	Q_{act} (m^3/s)	Re_D	Manometer reading, h_m (mm Hg)	Manometer reading, h_w (mm water)	Q_{th} (m^3/s)	C_d (expt)	C_d(ISO)
1									
2									
3									
4									
5									

Orificemeter:

Supply pipe diameter = 21mm

Orificemeter inlet (D) and throat (d) diameter = 20mm and 14mm

Sr. No.	Volume, V (litres)	Time, t (sec)	Q_{act} (m^3/s)	Re_D	Manometer reading, h_m (mm Hg)	Manometer reading, h_w (mm water)	Q_{th} (m^3/s)	C_d (expt)	C_d(ISO)
1									
2									
3									
4									
5									

E 2: IMPACT OF JETS

AIM: To study the impact of jets on Stationary surfaces.

EXPERIMENTAL SETUP: Chamber, flat & hemispherical vane, nozzle, sliding weight, balance weight, pump, sump tank, measuring tank.

THEORY: When the flow of fluid is obstructed by the surface, there is change in momentum of the fluid. By Newton's 2^{nd} law, this change in momentum results in external force on the fluid. Again, by Newton's 3^{rd} law, the same force acts on the obstructing surface.

Consider a jet of water coming out from the nozzle, striking a flat plate. The jet after striking the plate will get deflected through 90°. Hence, the component of the velocity of jet in the direction of jet, after striking will be zero.

$$F_x = Rate\ of\ change\ of\ momentum\ in\ the\ direction\ of\ force$$
$$F_x = (\rho A v) \times (v - 0)$$
$$F_x = \rho A v^2$$

For deriving the above equation, initial velocity minus the final velocity is taken and not final minus initial velocity. If the force exerted on the jet is to be calculated, then final minus initial velocity is taken. But if the force exerted by the jet on the plate is to be calculated, then initial minus final velocity is taken.

Similar procedure is to be repeated for the hemispherical vanes but here, the deflection angle is 180° and thus the final velocity is '-v'.

PROCEDURE: Fill up sufficient water in sump tank. Do the priming. Fix the required vane (flat or hemispherical) to the fixing rod. Fix the nozzle in Perspex box at center and close the top cover. Adjust the balance weight, so that vane fixing rod is in horizontal position. Open the bypass valve fully. Start the pump. Slowly close the bypass valve. The jet strikes the vane. Vane fixing rod will become unbalanced. Put the sliding weight over the rod and adjust its distance such that the rod is horizontal. Note down the balance weight and its distance from the pivot using scale. Close the discharge valve of measuring tank. Turn the funnel to measuring tank and measure the time for 5 liters. Repeat the procedure for other vane and nozzle.

SPECIFICATIONS:

Nozzle diameter: 6 mm

Vane: 1. Flat (deflection angle is 90°)

2. Hemispherical (deflection angle is 180°)

SPECIMEN CACULATIONS:

1. Actual discharge, $Q = \frac{V}{t}$ (m^3/sec)

2. Velocity of jet at point $v = \frac{Q}{A}$ (m/s)

3. Force exerted by vane :-
   (a) For **flat vane** (1), $F_{th} = \rho A v^2$ (N)

   (b) For **hemispherical vane** (2), $F_{th} = 2\rho A v^2$ (N)

4. Experimental force, $F_{expt} \times 0.135 = W \times L$

   (Moment about the hinge point)

   W =Weight of sliding weight N
   L = distance of sliding weight from fulcrum

5. Error (%) $Error\ (\%) = \frac{F_{th} - F_{expt}}{F_{th}} \times 100$

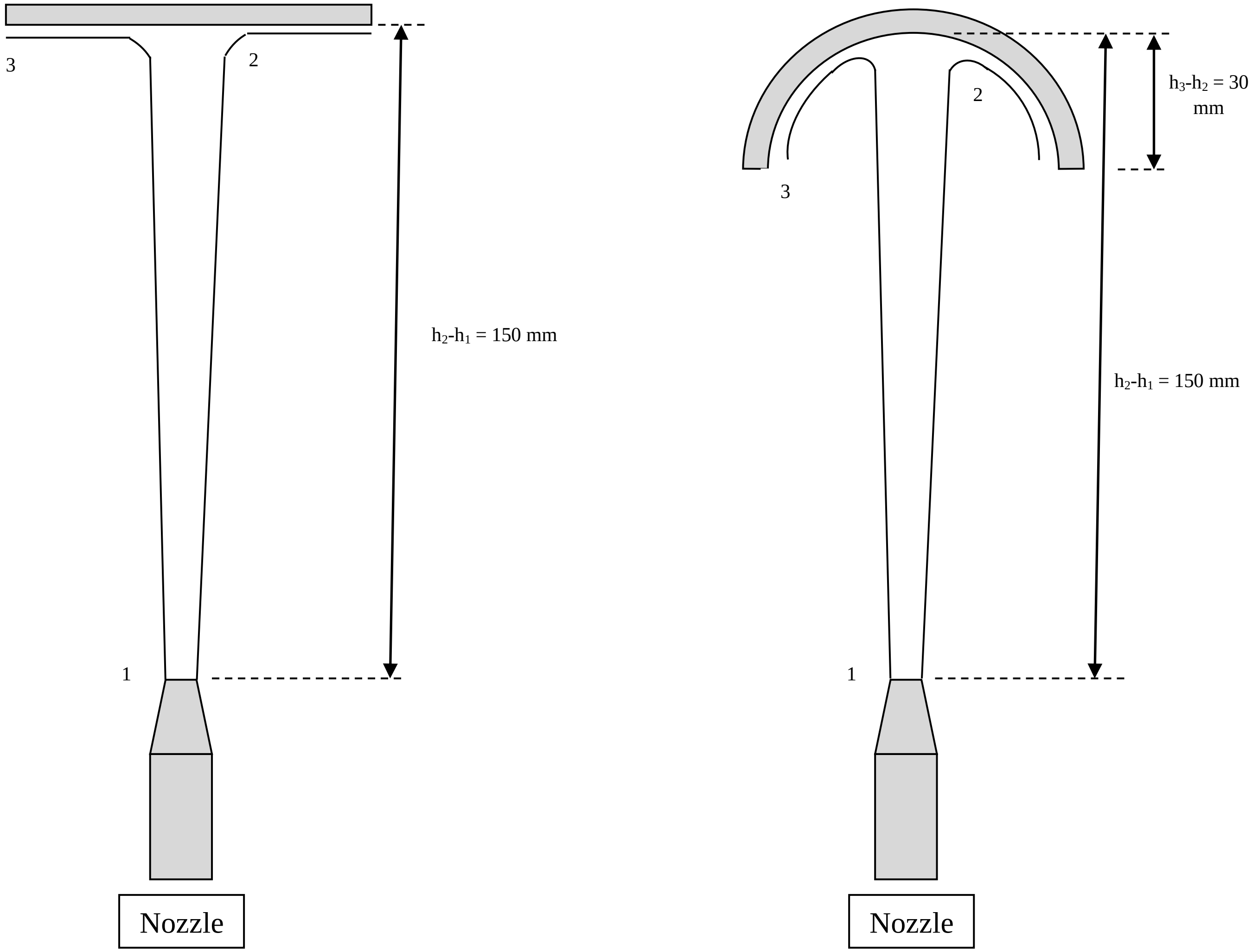


Figure 2.1: Jet nomenclature and dimensions

1 - Nozzle opening

2 - First contact point with the vane

3 - Last contact point with the vane

OBSERVATION TABLE:

Sr. No.	Nozzle diameter, d (mm)	Type of vane	Weight added, m (kg)	Distance of sliding weight, L (m)	Time for 5 litres rise in measuring tank, t (sec)	Discharge, Q (m^3/s)	Velocity, v (m/s)	Theoretical force, F_{th} (N)	Experimental force, F_{expt} (N)	Error (%)
1.	6	Flat vanes								
2.	6									
3.	6									
4.	6									
5.	6									
1.	6	Hemi-spherical vanes								
2.	6									
3.	6									
4.	6									
5.	6									

GRAPHS: Plot a graph of F_{th} & F_{expt} vs. velocity 'v'

RESULTS/DISCUSSIONS:

1. Write down the conclusions.
2. Try to explain the results from theory studied earlier.

SOURCES OF ERROR:

<u>E 3(a): LEVEL OF THE FLUID IN VORTEX FLOW</u>

AIM: To study the parabolic curve for forced vortex flow.

APPRATUS: Cylindrical vessels 300 mm diameter with central bottom outlet, mounted over rotating platform. D.C. motor with controller to rotate the sump tank, Measuring tank 300 × 300 × 300 mm mounted over the sump tank, Centrifugal pump to circulate the water, x-y co-ordinate measurement probe.

THEORY: When a liquid contained in a cylindrical vessel is forced to rotate either due to rotation of vessel about vertical axis or due to tangential velocity of water, surface of water no longer remains horizontal but it depresses at the centre and rises near the walls of the vessel.

The rotating mass of fluid is called **VORTEX** and motion of rotating mass of fluid is vortex motion. Vortices are of two types viz. forced vortex and free vortex. When a cylinder is rotated by external means then a vortex is called forced vortex. It is a case of rotational flow wherein the water molecules rotate about their own axis other than the revolution about the center of the tank. There is no shear in this case as this is a solid body rotation.

Apparatus consist of a Perspex cylinder with drain at center of bottom. The cylinder is fixed over a rotating platform, which can be rotated with the help of a D.C. motor at different speed. A tangential water supply rip is provided with flow control valve. The whole unit is mounted over the sump tank.

PROCEDURE: Close the drain valve of the cylinder vessel. Fill up some water (say 4-5 cm height from bottom) in the vessel. Switch ON the supply and slowly increase motor speed. Do not start the sump. Keep motor speed constant and wait till the vortex formed in the cylinder stabilizes. Once the vortex is stabilized, note down the co-ordinate of the vortex to complete the observation table. Also measure the rotation speed with the laser attachment of Tachometer.

SPECIMEN CALCULATION:

Rotational speed – N rpm

So angular velocity, $\omega = 2\pi N / 60$ rad/s

For forced vortex, $z = \left(\dfrac{\omega^2 r^2}{2g} \right)$ mm

Calculate different values of z at different values of$r$.

OBSERVATION TABLE:

In the experiment Perspex cylinder is rotating at ____RPM

Perspex cylindrical vessel diameter: 300 mm

Forced Vortex

Sr. No.	Radius (mm)	Experimental Height (mm)	Theoretical Height (mm)	RPM of the tank
1				
2				
3				
4				
5				
6				
7				
8				
9				
10				
11				
12				
13				
14				
15				
16				
17				
18				

GRAPHS: Plot the variation of *z* for different *r* for both experimental as well as theoretical for Forced vortex.

RESULTS /DISCUSSIONS:

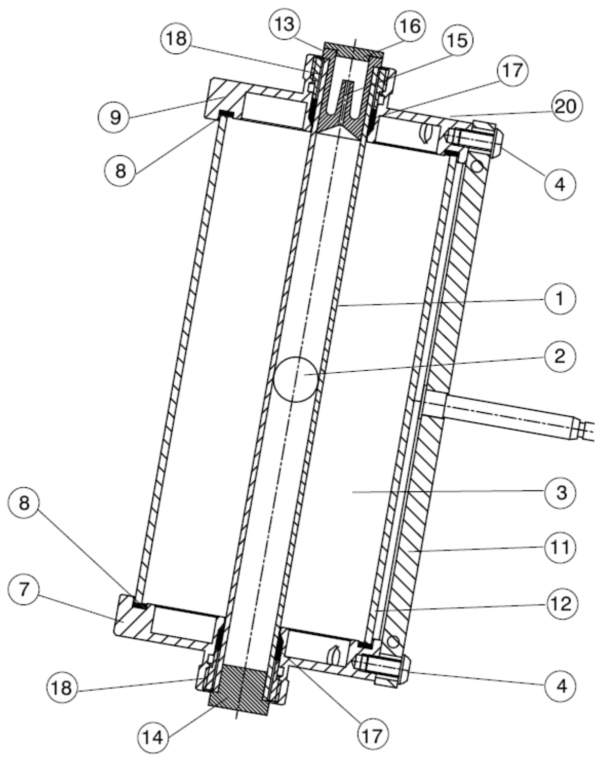
1. Write down the conclusions.
2. Try to explain the results from theory.
3. Write down the sources of errors. Calculate % error in experimental values.
4. Calculate theoretical angular velocity and show % deviation from experimental value.

E 3(b): FALLING BALL VISCOMETER

AIM: To determine the kinematic viscosity of a given sample of liquid using Falling Body Viscometer.

APPARATUS: Falling body viscometer set, stop watch.

THEORY: Falling Ball Viscometer measures the viscosity of transparent Newtonian liquids. This viscosity is correlated to the time a ball takes to fall a defined distance.



1 Falling tube
2 Ball
3 Tempering room
4 Screw
7 Set screw
8 Gasket
9 Cover
11 Brace
12 Jacket tube
13 Hollow stopper
14 Stopper
15 Capillary
16 Closing plate
17 Gasket for falling tube
18 Threaded bush
20 Connecting rod

Figure 3.1: Functional elements

The heart of the instrument is the measuring tube made of glass and a ball. This tube carries two ring marks A and B, which are spaced 50 cm apart. The measuring tube is jacketed by means of an outer glass tube, which encloses a room (space) to be filled with a temperature-controlled liquid. The measuring tube is closed on both sides by two stoppers, one of which contains a capillary and a small reservoir.

PROCEDURE:

The ball is set to the measuring position. The falling time of the ball moving from the ring mark A to ring mark B is determined by using a stop watch. The time period starts when the lower periphery of the ball touches the ring mark A, which must appear as a straight line.

The falling time ends when the lower periphery of the ball touches the ring mark B, which again must appear as a straight line. It is good practice to take the mean value out of several falling time values (2 to 3).

SPECIFICATIONS:

Density of the fluid (1): ______________ kg/m^3
Density of the fluid (2): ______________ kg/m^3

CALCULATIONS:

1. From the force balance, we have

   $$W - D - F_B = 0$$

   Where, W = mg = weight of the ball,
   D = drag force,
   F_B = buoyancy force.

2. Buoyancy force,

   $$F_B = \frac{\rho_f g \pi d^3}{6}$$

   Where, ρ_f = density of the fluid and
   d = ball diameter.

3. Drag Force

   It is approximated by Stokes law for (Re<<1) as follows:

   $D = 3\pi\mu U d$

   Where, μ = dynamic viscosity of the fluid and
   U = terminal velocity.

4. Drag Coefficient

   $$D = \frac{1}{2} C_D \, A \, \rho_f U^2$$

   Finally using the viscosity, calculate the Reynolds number for each sphere.

GRAPH: Plot C_D vs *Re*

RESULTS/DISCUSSIONS:

SOURCES OF ERROR:

OBSERVATION TABLE:

Distance travelled = 0.5 m

Sr. No.	Diameter of ball, d (mm)	Mass of ball, m (kg)	Buoyancy force, F_B (N)	Time, t (sec)	Velocity of ball, U (m/s)	Dynamic viscosity (Pa s)	Kinematic viscosity (m^2/s)	Re	C_D
1.									
2.									
3.									
4.									
5.									
6.									
7.									
8.									
9.									
10.									

E 4: CHARACTERISTICS OF A SUBMERGED JET

AIM: To study the velocity decay characteristics of a submerged jet.

APPARATUS: Air jet issued from a nozzle, Pitot tube and a manometer.

THEORY: A submerged jet issuing into a surrounding medium entrains the ambient fluid and spreads in laterally as it moves along. As the jet imparts momentum to the surrounding fluid, which is entrained, the average velocity decreases continuously.

Jets can be defined as a pressure driven unrestricted flow of a fluid into a quiescent ambience. Since, a fluid boundary cannot sustain a pressure difference across it, the subsonic jet boundary is a free shear layer in which the static pressure is constant throughout. The boundary layer at the exit of the device develops as a free shear layer, mixing with the ambient fluid thereby entraining the ambient fluid in the jet stream. Thus, the mass flow at any cross section of the jet progressively increases thereby the jet spreads along the downstream direction. In order to conserve momentum, the jet centerline velocity decreases with downstream distance.

Zone 1 (Convergent zone): This region is called the potential core of the jet where the centerline velocity is equal to the nozzle outlet velocity. This region normally extends up to 4d to 6d, where d is the diameter of the nozzle exit.

Zone 2 (Transition zone): It is the region in which the centerline velocity starts to decay. The velocity decay can be approximated as proportional to $x^{-0.5}$, where x is the axial distance. This usually corresponds to a region from 6d to 20d, and it is known as the interaction region where shear layers from both sides merge.

Zone 3 (Self-Similar zone): In this region transverse velocity profiles are similar at different values of x and the centerline velocity decay is approximately proportional to x^{-1}.

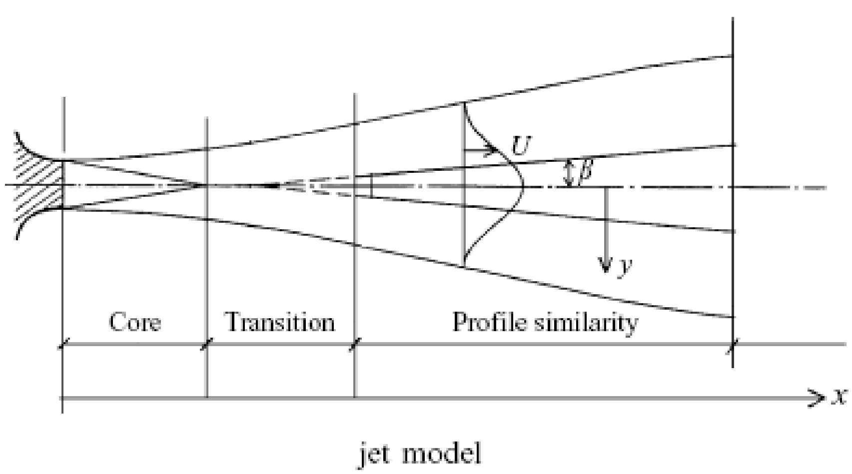


Figure 4.1: Typical velocity profile of the submerged jet (Malmstrom et al. 1997)

Since the jet spreads at constant pressure, there are no bounding walls; the momentum flux at any cross section (at a given distance along the jet axis) remains a constant. For the submerged jet, therefore, the cross-section of the jet increases, velocity profile becomes non-uniform and a part of the surrounding fluid is entrained in the main flow.

$$Discharge,\ Q = \int_0^R 2\pi r\, u\, dr$$

$$Momentum,\ M = \int_0^R 2\pi r\, \rho u^2\, dr$$

Momentum remains constant in the absence of external forces,

$$Energy,\ E = \int_0^R 2\pi r\, \rho u^3\, dr$$

Energy may drop due to losses due to the entrainment action.

PROCEDURE:

Part 1

Traverse the pitot probe along the centerline of the submerged jet and take manometer readings for centerline velocity decay at axial locations from 0 to 20*d* in steps of 2*d* (where *d* = nozzle exit diameter) from the nozzle.

The centerline velocity decay in axis-symmetric jets is typically modeled by simple decay equation:

$$\frac{U}{U_o} = K\frac{d}{Z}$$

U – Velocity at a given axial location (m/s)
U_o – Velocity at the exit of the nozzle (m/s)
d – Nozzle diameter (mm)
Z – Axial location (mm)

$$\frac{U}{U_o} = m\left(\frac{1}{Z/d}\right) + c\ ;\ Let\ \frac{U}{U_o} = Y\ and\ X = \left(\frac{1}{Z/d}\right)$$

OBSERVATION TABLE:

Diameter of the jet, d = 7.5 mm

PART 1:

Sr. No. k	Z/d	h (mm of water)	U (m/s)	1/(Z/d) [X]	U/Uo [Y]	[XY]	[X^2]
1	0						
2	2						
3	4						
4	6						
5	8						
6	10						
7	12						
8	14						
9	16						
10	18						
11	20						
				$\sum_{k=1}^{N} X_k =$	$\sum_{k=1}^{N} Y_k =$	$\sum_{k=1}^{N} X_k Y_k =$	$\sum_{k=1}^{N} X_k^2 =$

$$m = \frac{(N\sum_{k=1}^{N} X_k Y_k - \sum_{k=1}^{N} X_k \sum_{k=1}^{N} Y_k)}{(N\sum_{k=1}^{N} X_k{}^2 - (\sum_{k=1}^{N} X_k)^2)}$$

$$C = \frac{(\sum_{k=1}^{N} X_k{}^2 \sum_{k=1}^{N} Y_k - \sum_{k=1}^{N} X_k \sum_{k=1}^{N} X_k Y_k)}{(N\sum_{k=1}^{N} X_k{}^2 - (\sum_{k=1}^{N} X_k)^2)}$$

$$U = \sqrt{\frac{2\,\Delta P}{\rho_{air}}} \qquad \Delta P = \rho_w\, g\, h_w$$

PART 2:

Traverse the Pitot tube laterally from the centre to outward for velocity profile at locations 8*d* and 14*d*. Take the readings at an interval of r =1 mm.

At 8*d*:

Sr. No.	*r* *(mm)*	*h* (mm of water)	*U* (m/s)	U^2	r^2
1	0				
2	1				
3	2				
4	3				
5	4				
6	5				
7	6				

At 14*d*:

Sr. No.	*r* *(mm)*	*h* (mm of water)	*U* (m/s)	U^2	r^2
1	0				
2	1				
3	2				
4	3				
5	4				
6	5				
7	6				

Method of least squares to calculate *m* and *C*

$\boldsymbol{Y = mX + C}$ is the fit to (X_k , Y_k) experimental points

Error = $\boldsymbol{E = \sum\left[\left(mX_k + C\right) - Y_k\right]^2}$

Standard deviation = $\sigma^2 = \frac{1}{N\text{-}1}\sum_{k=1}^{N}\left[\left(mX_k + C\right) - Y_k\right]^2$

This standard deviation would possibly be the same if one repeated the "actual point – measured point" calibration over and over again. Assuming same standard deviation would occur even if a particular reading were to be repeated over and over again.

For 99.7% confidence level i.e. 3σ, the actual value = (mX + C) ± 3σ.

GRAPHS: At *8d*&*14d*, plot (1) $\boldsymbol{U}$ vs. $\boldsymbol{r^2}$ and (2) $\boldsymbol{U^2}$ vs. $\boldsymbol{r^2}$

Discharge at each location is given by- $\pi \times$ (area under the graph u vs. r^2). Similarly, momentum is given by- $\pi\rho \times$ (area under the graph u^2 vs. r^2) and energy is given by- $\pi\rho\times$ (area under the graph u^3vs. r^2). Calculate the discharge and momentum at both locations.

RESULTS/DISCUSSION:

1. Write down the conclusions.
2. Try to explain the results from theory

SOURCES OF ERROR:

E 5: PRESSURE DISTRIBUTION FOR FLOW AROUND A CIRCULAR CYLINDER

AIM: To study the pressure distribution for flow around a circular cylinder and compare it with the theoretical predictions.

APPRATUS: Circular cylinder, static pressure probe fitted in the cylinder, static pressure tapping at the test section, U–tube manometers, protractor for angular measurement, variable speed blower etc.

THEORY: The pressure distribution around the circular cylinder based on the ideal fluid flow theory is expressed by pressure coefficient C_p given by:

$$C_p = \frac{P - P_\infty}{\rho V_\infty^2 / 2} = 1 - 4\,(\sin\theta)^2$$

Where, P is the pressure on the surface of the cylinder at any angle θ, P_∞ is the upstream static pressure and V_∞ is the upstream velocity of flow.

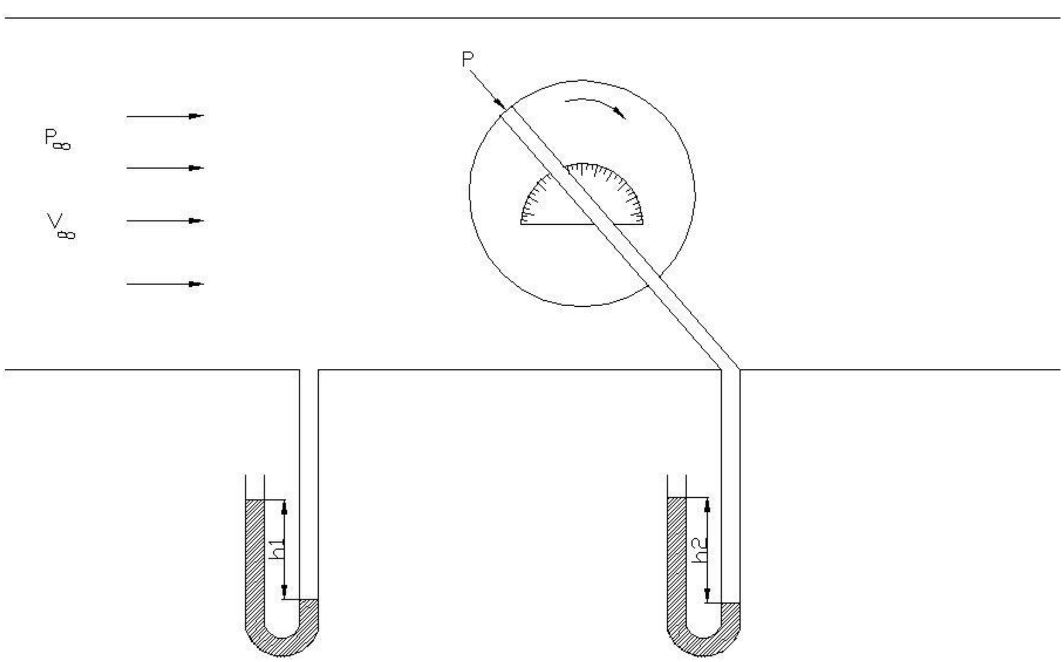


Figure 5.1: Schematic of the test setup

In the viscid region (near the solid boundary), up to θ = 90°, the viscous force opposes the combined pressure force and the force due to acceleration. Fluid particles overcome this viscous resistance due to continuous conversion of pressure force into kinetic energy. Beyond θ = 90°, within the viscous zone, the flow structure becomes different, the force due to acceleration is opposed by both the viscous force and pressure force. Due to higher momentum in case of turbulent flow, the boundary layer separates farther away than in the case for laminar flow.

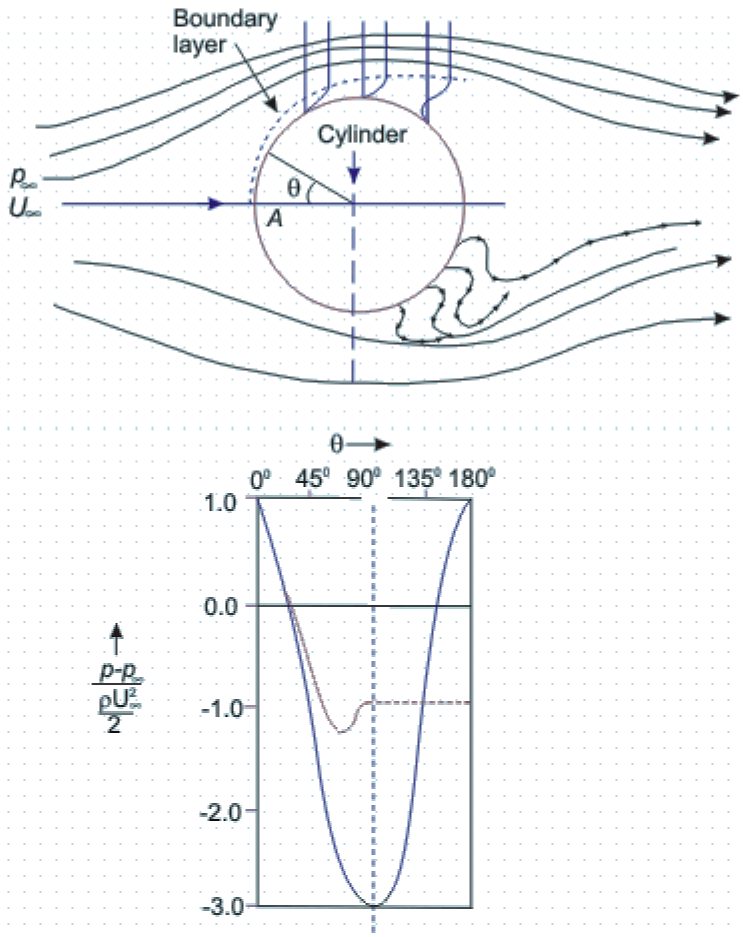


Figure 5.2: Flow around a cylinder

(https://nptel.ac.in/courses/112104118/lecture-29/29-4_seperation.htm)

PROCEDURE: Set a particular flow rate of air from the blower by adjusting the rheostat. Set the static pressure probe at θ = 0-degree position. Note down deflections h_1 and $(h_2)_{\theta = 0}$. Rotate the cylinder and note down deflections h_2 at various positions of the static pressure probe between 0° to 180° for every 5° rotation of the cylinder for one flow rate. Tabulate the readings and draw a neat schematic sketch of the experimental set up.

SPECIMEN CALCULATIONS:-

1. Average velocity,

$$V = \sqrt{\frac{2\,(P_\theta - P_\infty)}{\rho_{air}}}$$

2. Reynolds Number

$$Re = \frac{\rho\, V_{\infty\, D}}{\mu}$$

3. Coefficient of pressure (C_p)

Experimental, $$C_p = \frac{P - P_\infty}{\rho V_\infty^2 / 2}$$

Theoretical, $$C_p = 1 - 4\,(\sin\theta)^2$$

GRAPHS: Plot the variation of $C_{p,expt}$ with angle for various Reynolds numbers and compare the result with theoretical pressure coefficient $C_{p,theoretical}$.

RESULTS/DISCUSSION:

1. Write down the conclusions.
2. Try to explain the results from theory

SOURCES OF ERROR:

OBERVATION TABLE:

Diameter of the cylinder, D	0.008 m
Density of air at room temperature, ρ_{air}	1.2 kg/m^3
Kinematic viscosity of air at room temperature, ν	0.000016 m^2 / s
Density of water at room temperature, ρ_{water}	1000 kg / m^3

Smooth Cylinder:

Sr. No.	Angular Position θ (degrees)	$h_2 - h_1$ (m)	V (m/s)	C_p Experimental	C_p Theoretical
1					
2					
3					
4					
5					
6					
7					
8					
9					
10					
11					
12					
13					
14					
15					
16					
17					
18					
19					
20					
21					
22					
23					
24					
25					

Rough Cylinder:

Sr. No.	Angular Position θ (degrees)	$h_2 - h_1$ (m)	V (m/s)	C_p Experimental	C_p Theoretical
1					
2					
3					
4					
5					
6					
7					
8					
9					
10					
11					
12					
13					
14					
15					
16					
17					
18					
19					
20					
21					
22					
23					
24					
25					

E 6: EFFICIENCY OF A SQUARE CURVED DIFFUSER

AIM: To determine the pressure recovery efficiency of square curved diffuser of various geometries.

APPARATUS: Square curved diffusers, Pitot tube arrangement with water manometers, thermocouples.

THEORY:

A subsonic diffuser is a diverging flow passage where velocity of fluid particles decreases and pressure increases in the direction of flow. The diffusers find application in turbomachines, Venturimeter, wind tunnels and many other fluid handling systems.

Energy losses in a diffuser can be divided into two parts: Frictional losses and expansion losses. Theoretically, the pressure will increase as the area ratio (A_2 / A_1) increases but at higher area ratios, fluid particles have tendency to separate from the boundary of the diffuser resulting in high eddy losses, reducing the pressure recovery.

As such, for a given inlet condition, there is an optimum geometry of the diffuser for efficient pressure recovery. The pressure recovery efficiency is the ratio of actual pressure rise to the theoretical pressure rise and is given by-

$$\eta_P = \frac{\frac{P_2}{\rho_2} - \frac{P_1}{\rho_1}}{\frac{V_1^2 - V_2^2}{2}} = \frac{\frac{V_1^2 - V_2^2}{2} - Losses}{\frac{V_1^2 - V_2^2}{2}}$$

Theoretically the pressure rise will increase as the area ratio (A_2 / A_1) increases. However, at higher area ratios, fluid particles have a tendency to separate from the boundary of the diffuser resulting in high separation losses and reduction in pressure recovery.

The various factors which influence the diffuser performance are area ratio, angle of divergence, entrance conditions of flow, exit conditions of flow, shapes of the walls and wall roughness.

PROCEDURE:

Note down the geometrical parameters (inlet area, outlet area) of the diffuser which is connected to the experimental setup. Adjust a particular flow rate for the blower and note down the reading h_{max} (in terms of height of water) of the Pitot tube fixed at the center of the duct. Note down the differential pressure of the diffuser P_2-P_1 from the water manometer. As the diffuser is discharging into the atmosphere, the outlet pressure P_2 is atmospheric. Note down h_{max}, ΔP, T_1

and T_2 for at least 5 different flow rates by operating the regulating plate at the suction of the blower.

OBSERVATION TABLE:

Atmospheric Pressure	101325 Pa
Inlet area of diffuser	40 mm × 40 mm
Outlet area of diffuser	415 mm×40 mm
Particular gas constant, *R*	287 J/kg. K
Kinematic viscosity of air, υ	15.68 x 10^{-6} m^2/s
Density of water, ρ_{water}	1000 kg/m^3

PART 1: Calculation of pressure recovery efficiency of a given diffuser

Sr. No.	Pitot tube reading, h_p (mm of water)	Manometer reading, h_w (mm of water)	P_2 (Pa)	P_1 (Pa)	U_{max} (m/s)	V_1 (m/s)	V_2 (m/s)	*Re*	Pressure recovery efficiency η_p
1									
2									
3									
4									
5									

PART 2: Pressure distribution along the given diffuser

	Pressure Tap No.	Manometer reading (mm of water)	*ΔP* (Pa)
Upper edge	1		
	2		
	3		
	4		
Lower edge	1		
	2		
	3		
	4		

SPECIMEN CALCULATIONS:

$P_2 = \rho_w \, g \, h_w$ $\qquad\qquad$ $P_1 = 0$ *(Atmospheric)*

$U_{max} = \sqrt{\frac{2\,\Delta p}{\rho}}$; $\qquad\qquad$ $\Delta p = \rho_w \, g \, h_p$

$V_1 = 0.8\, U_{\max}$ *(Assuming turbulent flow- check whether the flow is turbulent or not)*

$$Re = \frac{V_1 D_1}{\nu_1},\ \ where\ D_1 = mean\ hydraulic\ diameter = \frac{4 \times c/s\ Area\ at\ inlet}{Wetted\ perimeter\ at\ inlet};$$

Pressure recovery efficiency, $\eta_P = \dfrac{P_1 - P_2}{\frac{\rho\,(V_1^2 - V_2^2)}{2}}$

GRAPHS: Plot η_p vs. *Re*.

RESULTS/DISCUSSIONS:

SOURCES OF ERROR:

E 7: FRICTION FACTOR IN INTERNAL PIPE

AIM: To experimentally study the frictional losses in pipe.

EXPERIMENTAL SETUP: Four pipes of different diameters and different material, control valves, sump tank U-tube manometers.

THEORY: When a fluid flows through a pipe, it is subjected to resistance due to shear forces between fluid and wall and also between fluid layers. This resistance is called as 'frictional resistance'. This resistance depends on various factors such as flow velocity, type of flow (laminar or turbulent), wall surface conditions etc.

PROCEDURE: Fill the sump tank with sufficient clean water. Open the outlet valve of pump and start the pump. Open the outlet valve of pipe to be tested. Remove all the air bubbles from manometer and connecting pipe. Adjust the flow such that the manometer head is readable.

Note down the manometer head and flow rate. Now increase the flow (operate outlet valve also so that there is no overflow) and take readings. Repeat the same procedure for other pipes.

SPECIFICATIONS:

Four pipes:

G.I. (Galvanized iron) pipe with internal diameter (I.D.) 21 mm
G.I. pipe with I.D. 17 mm
Copper pipe with I.D. 14.5 mm
Aluminum pipe with I.D. 12.5 mm

Test length of pipe = L = 1 m

SPECIMEN CACULATIONS:

1. Discharge: Q = Volume/Time (m^3/s)

2. Velocity of flow: $V = \frac{Q}{A}$ (m/s)

Area, $A = \frac{\pi}{4} \times D^2$ m^2 (D = internal diameter of pipe)

3. According to Darcy-Weisbach equation,

$$h_f = \frac{fLV^2}{2gD}$$

Where, f = friction factor

Then, $$f = \frac{2gDh_f}{LV^2}$$

4. According to Von-Karman correlation

$$\frac{1}{\sqrt{f}} = -2.0\,log_{10}\left[\frac{\varepsilon/D}{3.7}\right]$$

GRAPHS: Plot a graph of friction factor, *f* (expt.) vs. *Re*.

RESULTS/DISCUSSIONS:

1. What is the roughness of the pipe?
2. Write down the conclusions.

SOURCES OF ERROR:

Useful Data:

Density of mercury = 13600 kg/m^3
Density of water = 1000 kg/m^3
Viscosity of water, μ = 0.0007975 Pa-s

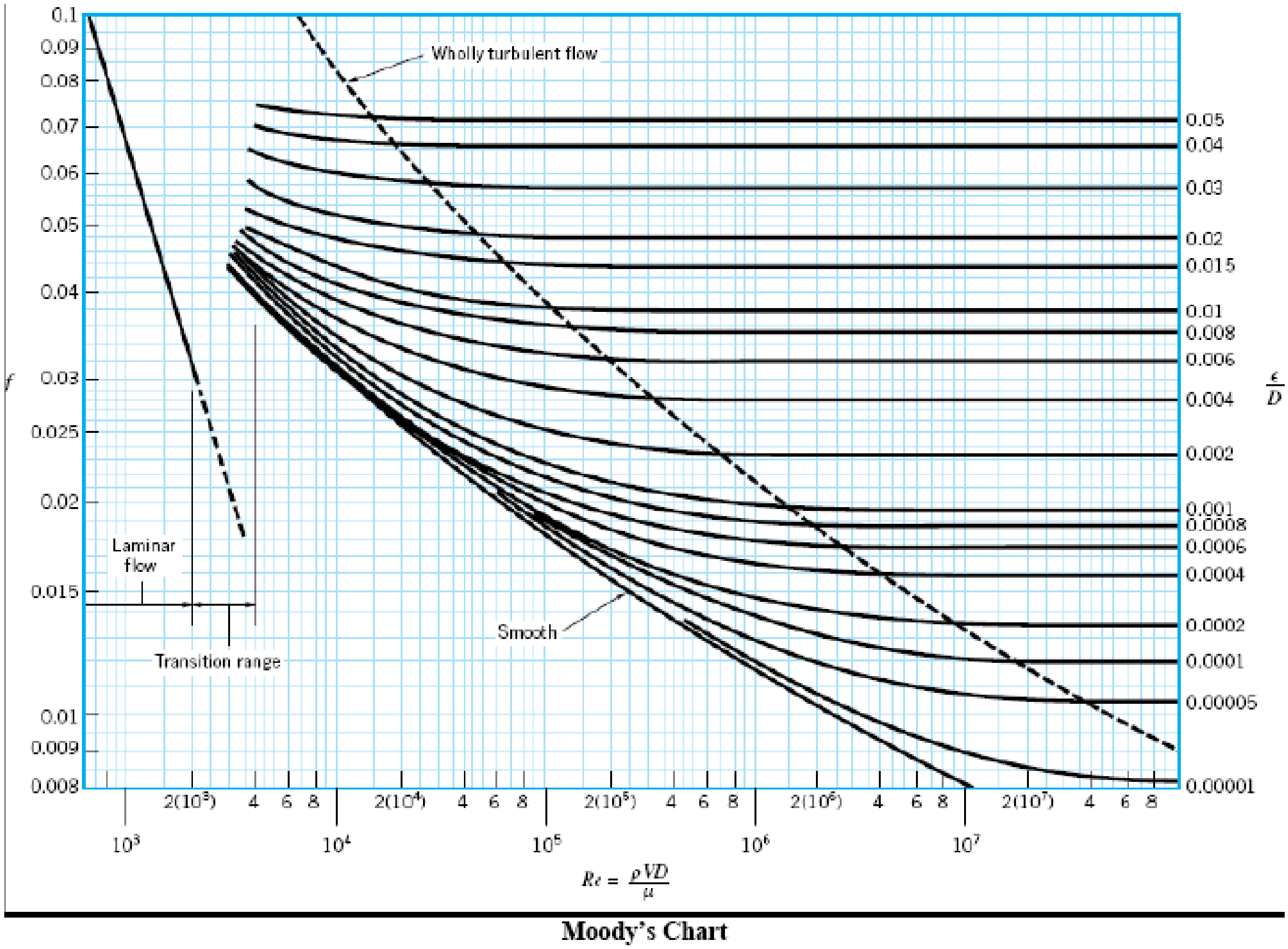


Figure 7.1: Moody's Chart

OBSERVATION TABLE:

Pipe	Sr. No.	Manometer reading 'h_m'(mm water)	Time for 5 litres to fill 't' (sec)	Discharge 'Q' (m^3/sec)	Velocity 'V' (m/s)	f (expt)	Re	ε/D (Von-Karman's relation)	ε/D (Moody's chart)
21 mm G.I.	1.								
	2.								
	3.								
	4.								
17 mm G.I.	1.								
	2.								
	3.								
	4.								
14.5 mm Cu	1.								
	2.								
	3.								
	4.								
12.5 mm Al	1.								
	2.								
	3.								
	4.								

E 8: LOSSES IN PIPE FITTINGS

AIM: To experimentally study the head losses in various pipe fittings such as elbow, bend, sudden expansion, sudden contraction etc.

EXPERIMENTAL SETUP: Fittings- bend, elbow, sudden expansion, sudden contraction etc., pressure tapping at inlet & outlet of each fitting, common differential manometer, pump, bypass valve to control water flow.

THEORY: While installing the pipeline to transfer a fluid, it is not possible to install a long straight pipe of same size throughout due to space restriction, location of outlet etc. Hence, we use various fittings. All these fittings cause head loss.

Bend/Elbow: Loss of head occurs due to change in direction.

Sudden Expansion: The fluid flows against an adverse pressure gradient. The upstream pressure p_1 at section a-b is lower than the downstream pressure p_2 at section e-f since the upstream velocity V_1 is higher than the downstream velocity V_2 as a consequence of continuity. The fluid particles near the wall due to their low kinetic energy cannot overcome the adverse pressure hill in the direction of flow and hence follow up the reverse path under the favorable pressure gradient (from p_2 to p_1). This creates a zone of recirculating flow with turbulent eddies near the wall of the larger tube at the abrupt change of cross-section, as shown in the figure, resulting in a loss of total mechanical energy.

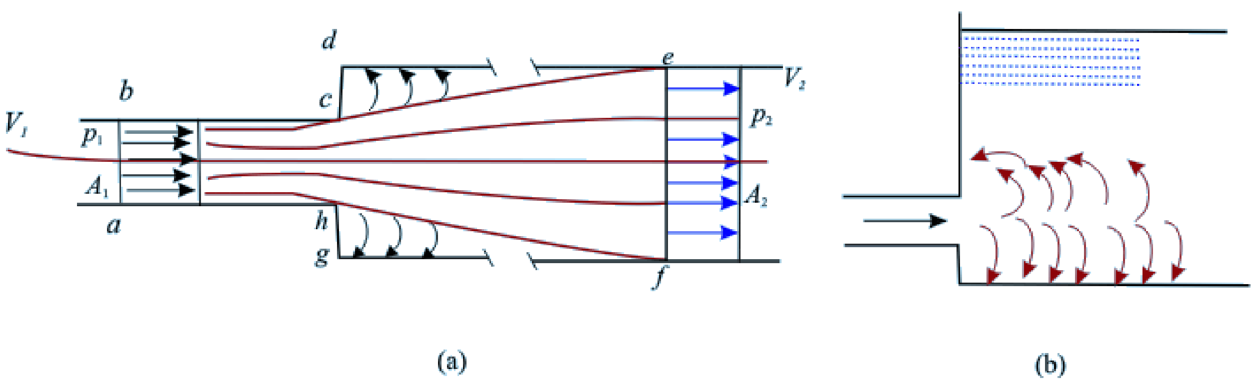


Figure 8.1: Sudden Expansion

(https://nptel.ac.in/courses/112104118/lecture-14/14-6_losses_sudden_enlarg.htm)

Sudden Contraction: An abrupt contraction is geometrically the reverse of an abrupt enlargement. The streamlines cannot follow the abrupt change of geometry and hence gradually converge from an upstream section of the larger tube. However, immediately downstream of the junction of area contraction, the cross-sectional area of the stream tube becomes the minimum and less than that of the smaller pipe. This section of the stream tube is known as vena-contracta, after which the stream widens again to fill the pipe. The velocity of flow in the converging part of the stream tube from Sec. 1-1 to Sec. c-c (vena contracta) increases due to continuity and the pressure decreases in the direction of flow accordingly in compliance with the Bernoulli's theorem. In an accelerating flow, under a favorable pressure gradient, losses due to separation cannot take place. But in the decelerating part of the flow from Sec. c-c to Sec. 2-2, where the stream tube expands to fill the pipe, losses take place in the similar fashion as occur in case of a sudden geometrical enlargement. Hence eddies are formed between the vena contracta c-c and the downstream Sec. 2-2. The flow pattern after the vena contracta is similar to that after an abrupt enlargement, and the loss of head is thus confined between Sec. c-c to Sec. 2-2. Therefore, the losses due to contraction are not for the contraction itself, but due to the expansion followed by the contraction.

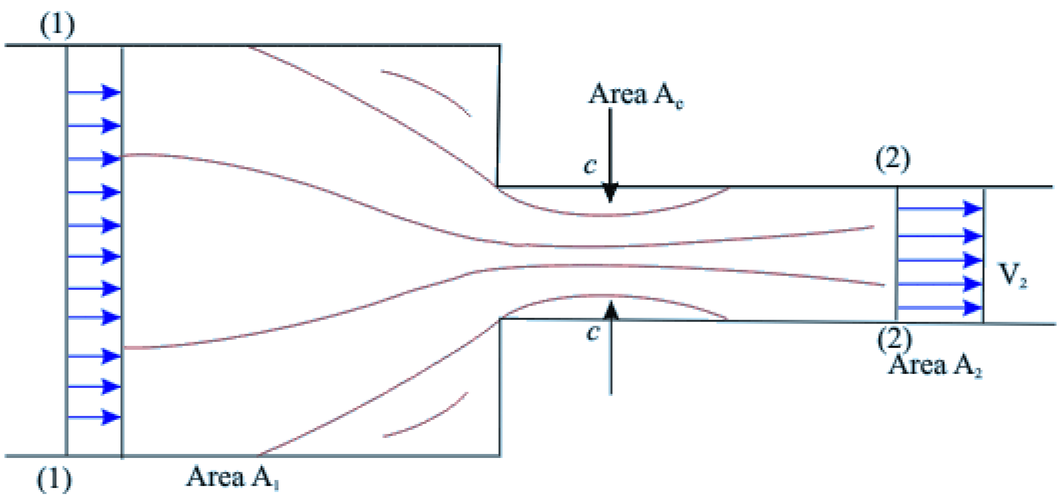


Figure 8.2: Sudden Contraction

(https://nptel.ac.in/courses/112104118/lecture-14/14-7_losses_sudden_contract.htm)

PROCEDURE: Fill the sump tank with sufficient clean water. Open the pump discharge valve & bypass valve fully. Keep manometer cocks closed. Start the pump and open the manometer connection of bend, operate both the cocks simultaneously. Ensure that there is no air bubble in the line. Note the manometer reading and flow rate, repeat the procedure for different flow rates. Close the manometer cocks. Connect other fitting to manometer and repeat the above procedure.

SPECIFICATIONS:

Basic pipe of diameter, d = 15 mm
Bend diameter, d_b = 19 mm
Elbow diameter, d_e = 14 mm
Sudden expansion from ∅ 16 mm to ∅ 27.5 mm
Sudden contraction from ∅ 27.5 mm to ∅ 16 mm
Density of water, ρ = 1000 kg/s
Viscosity of water, μ = 0.0007975 Pa-s
0.5 hp pump to circulate water

SPECIMEN CACULATIONS:

1. **Bend and Elbow:**

Mean area, $A = \frac{\pi}{4} \times d^2$ (based on the basic pipe diameter)

Mean velocity of flow, $V = \frac{Q}{A}$

$$Q = Volume/Time$$

$$Re = \frac{\rho\, V\, d}{\mu}$$

$$Loss\ Coefficient, K = \frac{\Delta p}{\frac{1}{2}\,\rho V^2} \qquad \Delta p = \rho_w\, g\, h_w$$

$$h_w = \left(\frac{\rho_m}{\rho_w} - 1\right) h_m$$

2. **Sudden expansion:**

$$\frac{p_1}{\rho g} + \frac{V_1^2}{2g} = \frac{p_2}{\rho g} + \frac{V_2^2}{2g} + h_e \qquad (Bernoulli's\ Principle)$$

$$h_e = \frac{p_1 - p_2}{\rho g} + \frac{V_1^2 - V_2^2}{2g} (p_1 > p_2), \quad (V_2 > V_1)$$

Mean velocity of flow, $V = \frac{Q}{A}$ (based on the basic pipe diameter)

$$K_e = \frac{g\, h_e}{V^2/2}$$

3. Sudden contraction:

$$h_c = 0.5 \frac{V_2^2}{2g}$$

Mean velocity of flow, $V = \frac{Q}{A}$ (based on the basic pipe diameter)

$$K_c = \frac{g\, h_c}{V^2/2}$$

RESULTS/DISCUSSION:

1. Write down the conclusions.
2. Try to explain the results from theory studied earlier.

SOURCES OF ERROR:

OBSERVATION TABLE:

Fitting	Sr. No.	Time for 5 litres rise in measuring tank 't' (sec)	Discharge 'Q' (m^3/sec)	Velocity 'V' (m/s)	Manometer reading 'h_m' (mm Hg)	Δp (Pa)	*Re*	*K*
Elbow	1							
	2							
	3							
	4							
Bend	1							
	2							
	3							
	4							

Fitting	Sr. No.	Time for 5 litres rise in measuring tank 't' (sec)	Discharge 'Q' (m^3/sec)	'V_1' (m/s)	'V_2' (m/s)	Manometer reading 'h_m' (mm Hg)	ΔP (Pa)	*Re*	*K*
Sudden expansion	1								
	2								
	3								
	4								
Sudden contraction	1								
	2								
	3								
	4								

E 9: TURBULENT VELOCITY PROFILE IN A CIRULAR PIPE

AIM: To study the velocity distribution for turbulent flow through circular pipe

APPARATUS: Pipes with pressure tapings, Pitot tube, mercury manometer, arrangement for traversing Pitot tube through the pipe, volumetric tank, stop watch etc.

THEORY: In turbulent flow, the adjacent layers mix intimately so that there are velocity fluctuations which are superimposed on the mean average flow. Due to this mixing of the fluid layers there is a momentum transfer in the direction normal to the flow resulting in a flatter velocity profile than that in laminar flow.

The energy loss for turbulent flow in a pipe depends upon the density of the fluid, velocity of the flow V, diameter of the pipe D, dynamic viscosity of the fluid, absolute roughness of the pipe and length of the pipe L. Dimensional analysis for pressure drop (P) in a straight circular pipe shows that P/V^2 is a function of Reynolds number, V, D, relative pipe roughness e/D and length to diameter ratio (L/D).

The pressure drop ΔP in pipe is related by Darcy - Weisbach equation:

$$\frac{\Delta p}{\gamma} = \frac{fLV^2}{2gD}$$

where, ' γ ' is the specific weight of fluid.

As such the friction factor *f* is a function of Reynolds number Re and e/D. Moody's diagram for pipe friction factor gives the variation of friction factor *f* with Reynolds number for various relative roughness e/D of the pipes. For most of the flows the fluid velocity is zero at the wall to satisfy no slip condition and is a maximum at the centerline of the pipe.

The way in which the velocity of fluid particles varies from zero at the wall to maximum at the center is a characteristic of that regime. For the turbulent flow regime, the velocity profile equation of the fluid particles follows the power law as given by-

$$\frac{V}{V_{max}} = \left(1 - \frac{r}{R}\right)^{\frac{1}{n}}$$

where,

V – Velocity of fluid particle at a particular point in the pipe cross-section,

V_{max} - velocity of the fluid particles at the centerline of the flow,

r - Radial distance of the fluid particle having velocity u from the centerline of the pipe

R- Radius of the pipeline.

The exponent n is a function of the Reynolds number Re of the flow. Thus, value of n and hence the Reynolds number Re of the flow, decides the velocity profile variation across the pipe cross-section.

Also, the exponent n relates the average and maximum velocity of the flow by the following correlation-

$$\frac{V_{avg}}{V_{max}} = \frac{2n^2}{(2n+1)(n+1)}$$

PROCEDURE:

For velocity profile determination traverse the pitot tube through the entire pipe cross-section to determine the position of the pitot tube coinciding with the centerline of the pipe. Traverse the pitot tube in the lower half of the pipe using the scale provided with the traversing arrangement. Note the stagnation pressure at all these radial positions of the pipe. Repeat the procedure for the upper half of the pipe.

SPECIMEN CALCULATIONS:

$\Delta P = \rho_m \, g \, h_m$

Velocity at a given point, $V = \sqrt{\frac{2\,\Delta P}{\rho_{air}}}$

RESULTS/DISCUSSIONS:

1. Write down the observations.
2. Try to explain the results from theory.

Fitting exponential curve by least squares:

$$y = a_0 x^{a_1}$$

$$\ln y = \ln a_0 + a_1 \ln x$$

$$Y_i = a_0 + a_1 X_i$$

$$D = \sum_{i=1}^{N} \left[Y_i - a_0 - a_1 X_i \right]^2$$

$$\frac{\partial D}{\partial a_0} = 0 \Rightarrow \quad 2\sum_{i=1}^{N}\left(Y_i - a_0 - a_1 X_i\right)\left(-1\right) = 0$$

$$\frac{\partial D}{\partial a_1} = 0 \Rightarrow \quad 2\sum_{i=1}^{N}\left(Y_i - a_0 - a_1 X_i\right)\left(-X_i\right) = 0$$

Solving above equations simultaneously for a_0 and a_1 ,

$$a_0 = \frac{\sum_{i=1}^{N} Y_i - a_1 \sum_{i=1}^{N} X_i}{N} \quad a_1 = \frac{N\sum_{i=1}^{N} X_i Y_i - \sum_{i=1}^{N} X_i \sum_{i=1}^{N} Y_i}{N\sum_{i=1}^{N} X_i^2 - \left(\sum_{i=1}^{N} X_i\right)^2}$$

OBSERVATION TABLE:

Diameter of the pipes, D — 50 mm
Density of air, ρ — 1.2 kg/m^3
Dynamic viscosity of air, μ — 1.7894×10^{-5} Pa-s

Sr. No.	Vernier scale reading, r (mm)	Manometer reading, h_m (mm of water)	Velocity V (m/s)	$ln(1-r/R)$ (X)	$ln(V/V_{max})$ (Y)	(XY)	(X^2)
1.							
2.							
3.							
4.							
5.							
6.							
7.							
8.							
9.							
10.							
11.							
12.							
13.							
14.							
15.							
				$\sum_{i=1}^{N} X_i =$	$\sum_{i=1}^{N} Y_i =$	$\sum_{i=1}^{N} X_i Y_i =$	$\sum_{i=1}^{N} X_i^2 =$

E 10(a): HAGEN'S EXPERIMENT

AIM: To study laminar and turbulent regimes in a circular pipe using Hagen's method and determine the type of flow by calculating the value of exponent 'n'.

EXPERIMENTAL SETUP: Pressure gauge (mm H_2O), pipe, sump tank, pump, measuring container, Constant head supply tank, dye tank, transparent tube dye injection system, sump tank, regulating valve, stop watch.

THEORY:

Whenever a fluid flows through a pipe, the flow is either laminar or turbulent. When fluid is flowing in parallel layers sliding one over another it is called ***'laminar flow'***.

On the other hand, when fluid particles flow in random directions i.e. there is no motion in layers, it is called ***'turbulent flow'***. Existence of these two flows was first demonstrated by Osborne Reynolds.

A German engineer G. H. L. Hagen proposed the following relation using his experimental data:

$$\Delta \boldsymbol{p} \propto \boldsymbol{v}^{\boldsymbol{n}}$$

Where, n = 1.0 - 1.2 (laminar flow)

n = 1.75 – 2 (turbulent flow)

PROCEDURE:

Fill the sump tank with sufficient clean water up to the mark. Open the pump discharge valve & bypass valve fully. Start the pump and adjust the water flow to different flow rates. Measure the pressure drop for a given flow rate in the pressure gauge provided.

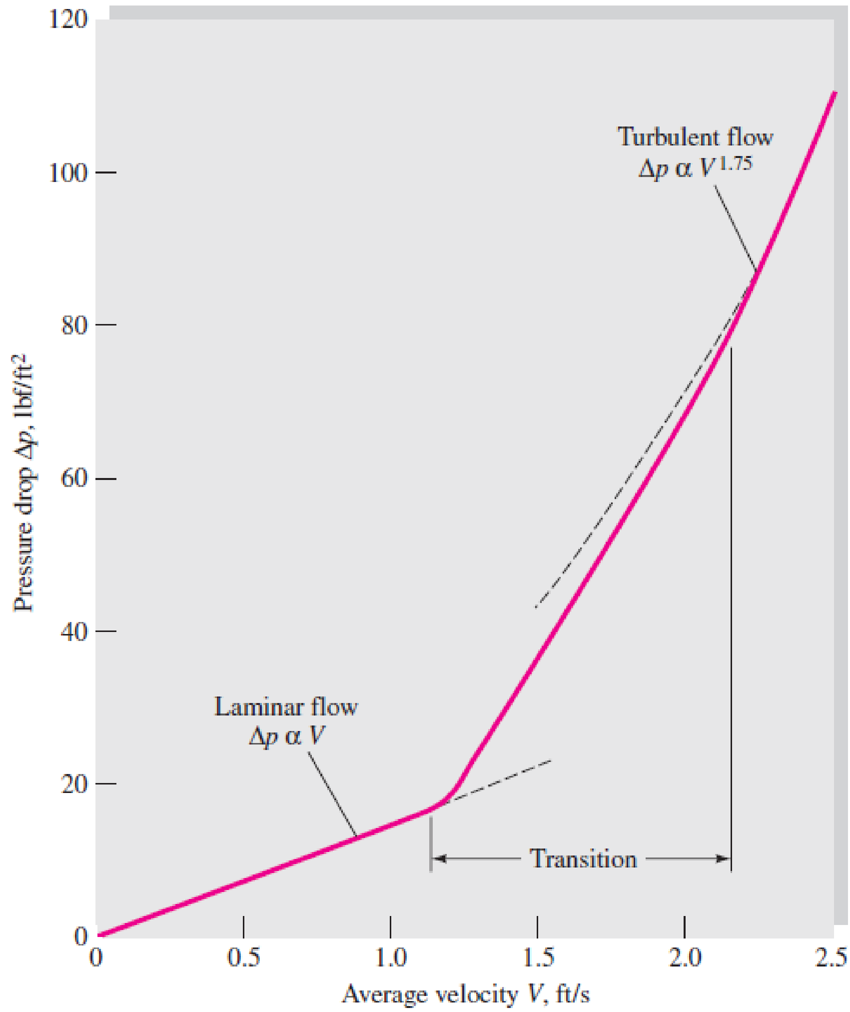
Reynolds Apparatus

Fill the sump tank with sufficient clean water up to the mark. Put sufficient dye of Potassium permanganate in dye tank. Open the pump discharge valve & bypass valve fully. Start the pump and adjust the water flow to a small rate.

Start the dye injection. Wait for some time. A steady dye line is observed. Slowly increase the flow rate, ensure that water level in supply tank is constant. At certain flow rate, the dye line will be disturbed, note down this flow rate using measuring flask and stopwatch. Further increase the flow, dye line diffuses over entire cross section, note down this reading also.

SPECIFICATIONS:

Internal diameter of pipe = 15 mm
Density of water = ρ = 1000 kg/m^3
Dynamic viscosity of water = μ = 0.0007975 N-s/m^2



Hagen's experiment

SPECIMEN CACULATIONS:

1. Discharge, $Q = Volume/Time$

2. Velocity of flow, $V = Q/A$
3. Reynolds number, $Re = \frac{\rho\, V\, L}{\mu} = \frac{V\, L}{\nu}$

Where, L = characteristic linear dimension.

ν = kinematic viscosity = $\dfrac{\mu}{\rho}$

OBSERVATION TABLE:

Sr. No.	Volume collected (litres)	Time to collect the volume 't' (sec)	Discharge 'Q' (m^3/sec)	Velocity 'V' (m/s)	Pressure drop Δ p (mm water)	*Re*	Type of flow
1.							
2.							
3.							
4.							
5.							
6.							
7.							

GRAPHS

Plot **'Δp'** vs **'V'** for Hagen's experiment and find the value of 'n' and check the type of flow.

RESULTS/DISCUSSIONS:

SOURCES OF ERROR:

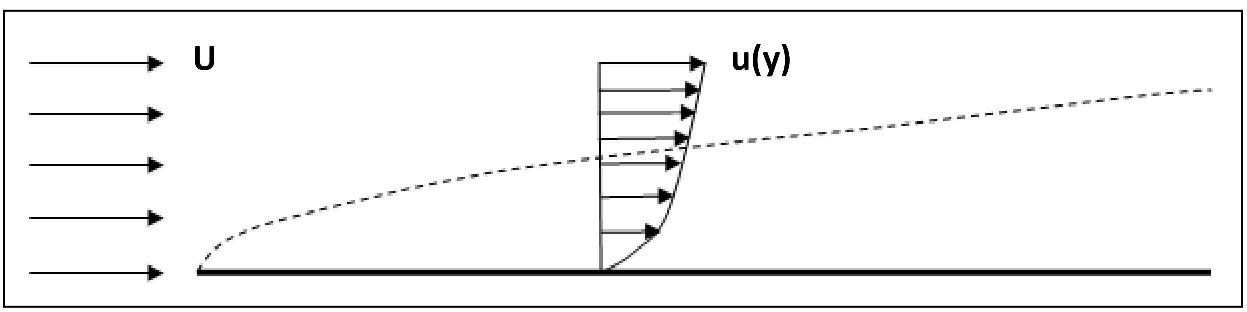
E 10(b): BOUNDARY LAYER THICKNESS MEASUREMENT

AIM: To measure the boundary layer thickness (δ) on a flat plate at different stream-wise locations for a given Reynolds number and compare the results with theory.

APPARATUS: Square channel, Blower, Flat plate, Pitot tube, manometer, Scale.

THEORY:

When a fluid flows past a body (e.g. cylinder or plate) or a wall, kept in the free stream of the flow, the fluid particles adhere to the boundary where the no-slip condition is satisfied. If the boundary is stationary, the velocity of fluid at the boundary will be zero. Farther away from the boundary, the velocity will be higher and as a result of this variation in velocity, a velocity gradient $\frac{du}{dy}$ will exist, where *y* is wall normal direction. The velocity of fluid increases from zero on a stationary boundary to free-stream velocity (U) of the fluid in the direction normal to the boundary. This takes place in a narrow region in the vicinity of solid boundary and this thickness is referred to as the boundary layer.



Boundary Layer Thickness (δ): It is the distance from the boundary of the solid body measured in the y-direction to the point, where the velocity of the fluid is approximately equal to 0.99 times the free stream velocity of the fluid.

Displacement Thickness (δ^*): It is the distance measured perpendicular to the boundary of the solid body, by which the boundary should be displaced to compensate for the reduction in flow rate on account of boundary layer formation.

$$\delta^* = \int_0^\delta \left(1 - \frac{u}{U}\right) dy$$

Laminar Boundary Layer

Assuming that the velocity profile takes the following form,

$$\frac{u(y)}{U} = \frac{3}{2}\left(\frac{y}{\delta}\right) - \frac{1}{2}\left(\frac{y}{\delta}\right)^3$$

Where, *u* is the fluid velocity at any particular location and *U* is the uniform free-stream velocity.

Applying the Von-Karman momentum integral equation, the boundary layer thickness can be theoretically found to be

$$\delta = \frac{4.64\,x}{\sqrt{Re}}$$

Using the differential equation, the Blasius solution for δ can be written as,

$$\delta = \frac{4.91\,x}{\sqrt{Re}}$$

Turbulent Boundary Layer

Blasius on the basis of experiment gave the following velocity profile for turbulent boundary layer:

$$\frac{u}{U} = \left(\frac{y}{\delta}\right)^n$$

Where, n = $\frac{1}{7}$ for $Re < 10^7$ but more than 10^5.

$$\delta = \frac{0.16\,x}{(Re)^{1/7}}$$

Here, we consider the critical Reynolds number (Re_{cr}) for transition from laminar to turbulent as $Re_{cr} = 2 \times 10^5$. Beyond Re_{cr}, the flow is considered as turbulent. The theoretical value for transition is between Re_{cr} = 3-5x10^5, however, this is under the assumption that the walls are smooth. In our experiment due to roughness and other effects the transition happens earlier.

PROCEDURE:

Measure the free stream velocity at the inlet, i.e., just before the starting of the flat plate.

For boundary layer thickness determination, traverse the Pitot tube from the bottom of the cross-section to the point where 99% of the value of free stream is attained. Determine this position of the Pitot tube (distance moved from the bottom in y- direction).

OBSERVATIONS:

Sr. No.	Location, x (cm)	Manometer reading (mm water)	B.L. thickness, δ (mm)	Re	δ_{th} (mm)	% Error
1.						
2.						
3.						
4.						
5.						
6.						

CALCULATIONS:

Dynamic viscosity of air (μ) = 1.846×10^{-5} Pa s

Density of air (ρ) = 1.2 kg/m^3

$$Re = \frac{\rho v L}{\mu}$$

For laminar flow,

$$\delta_{th} = \frac{4.64\,x}{\sqrt{Re}}$$

For turbulent flow,

$$\delta_{th} = \frac{0.16\,x}{(Re)^{1/7}}$$

GRAPHS: Plot the trend of the boundary layer thickness as a function of the stream-wise distance.

RESULTS/DISCUSSIONS:

1. Write the conclusions based on the observations and correlate with the theory studied.
2. Write down the sources of error.