

UG Fluid Mechanics Lab Manual

Department of Applied Mechanics,

Indian Institute of Technology, Delhi

Updated on July 31, 2025

Contents

1	Fully Developed Flow Through Pipes	2
2	Performance Characteristics of a Pelton Turbine	7
3	Conservation of Mechanical Energy (Bernoulli's Theorem)	11
4	Study of the Mean Flow of a Free Air (Submerged) Jet	16
5	Performance Characteristics of Free Jets	21
6	Obstruction Type Flow Meters	28
7	Performance Characteristics of a Centrifugal Pump	33
8	Flow Visualization	38
9	Flow Around a Circular Cylinder	43
10	Development of Turbulent Pipe Flow	49

Experiment 1

Fully Developed Flow Through Pipes

Objective:

To study the frictional characteristics of fully-developed laminar and turbulent pipe flow.

Practical Relevance:

Pipelines are used to transport fluids from one place to another. In the case of long pipelines, (e.g. those transporting oil and natural gas from offshore platforms to process plants on land) the frictional loss in the pipes constitutes a major energy loss. To design such pipelines and the pumping power required for them it is essential to have an understanding of frictional losses in laminar and turbulent pipe flow.

Theoretical Background:

The single most important factor in the study of fully-developed pipe flow is the Reynolds number ($Re = \rho U D / \mu$, where, U is the average velocity and D is the pipe diameter) which determines whether the flow is laminar or turbulent. For most pipe-flows, turbulence sets in when the Reynolds number exceeds 2000. The other important factor is the relative roughness

(ϵ/D , where, ϵ is the average roughness height), which determines the level of friction in turbulent flow. In a constant diameter pipe the energy loss over a length L , may be characterised by an equivalent head loss, h_f . The head loss per unit length, h_f/L , is proportional to the velocity for laminar flow while, for the turbulent flow, at large Re , h_f is proportional to the square of the velocity. One can also define a non-dimensional friction factor, f , as follows,

$$f = h_f \frac{2g}{U^2} \frac{D}{L} \quad (1.1)$$

It can be shown quite easily that for fully-developed laminar pipe flow the friction factor is given by,

$$f = \frac{64}{Re} \quad (Re < 2000) \quad (1.2)$$

No such theoretical relationship exists for turbulent flow, however, for smooth pipes ($\epsilon/D < 10^{-6}$), Blasius was able to fit the experimental data over a wide range of Reynolds numbers ($2000 < Re < 10^5$) with the following correlation,

$$f = \frac{0.316}{(Re)^{\frac{1}{4}}} \quad (2000 < Re < 10^5) \quad (1.3)$$

The variation of the friction factor with Re and the relative roughness, ϵ/D , is usually shown in the form of a Moody chart (see any text on basic engineering fluid mechanics).

Experimental set-up:

The rig consists of a constant-head supply tank connected to five pipes of different diameters by means of a common manifold, a volumetric tank at the common outlet, a sump and a pump. Each pipe is fitted with an inverted U-tube manometer to enable the head loss to be measured over a large distance L . Two of the five pipes have nearly the same diameter but one is a smooth pipe (brass) and the other is a rough pipe (G.I.). Thus the effect of roughness on the friction factor may be studied. A schematic diagram is shown in figure 1.

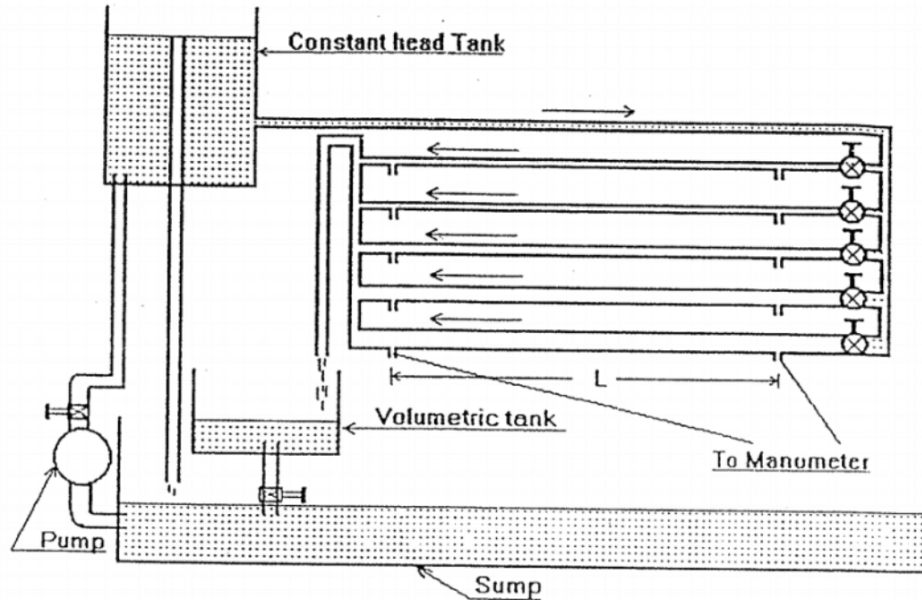


Figure 1.1: Schematic diagram of the set-up

Procedure:

1. Measure (note down) the pipe diameters for all the pipes to be studied.
2. Start the pump while keeping all five valves closed.
3. Choose one of the pipes and carefully open the valve on the input line to this pipe while keeping the other valves closed. Open the valve till the difference in the manometer levels is very large taking care that the fluid does not spill into the connecting tube.
4. Measure the pressure differential in the manometer provided.
5. Determine the flow rate using either the volumetric or the gravimetric method.
6. Repeat steps 6 for five more flow rates, progressively reducing the flow rate such that the pressure differential is approximately 2 cm of water for the lowest flow rate.

- Repeat steps 4 through 7 for the other four pipes.

Note: For the smallest diameter pipe calculate the head loss obtained for $Re = 1000$ from equations (1) and (2). Set this height h_f as the one for the lowest flow rate and then take 5 more readings till the highest flow rate is reached.

Observations:

- Cross-sectional area of the volumetric tank, $a = \text{--- m}^2$

Pipe No. 1:

- Pipe diameter, $D_1 = \text{--- m}$
- Observation Table:

Sr. No.	Manometer Reading h_f (water), m	Height filled in Volumetric Tank, h_v , m	Time taken for filling, t , s
1			
2			
3			
4			
5			
6			

- Analysis Table:

Sr. No.	Flow rate $Q = ah_v/t$, m^3/s	Actual Friction Factor f_{act} (eq 3.1)	Expected Friction Factor f (eq 3.2 or eq 3.3)	Reynolds Number Re
1				
2				
3				
4				
5				
6				

5. Repeat steps 2) through 4) for the other pipes.
6. Plot the following (for all the pipes together):
 - (a) $\log f_{\text{act}}$ vs. $\log \text{Re}$
 - (b) On the same graph draw lines corresponding to $f = 64/\text{Re}$ and $f = 0.316/(\text{Re})^{1/4}$.

Discussions:

1. Highlight what you consider to be the key results of the experiment and discuss the results with reference to the above graph.
2. List out the sources of errors and suggest methods for improving the experiment.

Experiment 2

Performance Characteristics of a Pelton Turbine

OBJECTIVE:

To obtain the performance characteristics of a Pelton turbine.

PRACTICAL RELEVANCE:

Turbines are devices which use the hydraulic energy of water and convert it into usable mechanical energy. The mechanical energy so developed is used to run an electric generator directly coupled to the shaft of the turbine. The electric power generated in this manner is cheaper than power developed by other sources such as coal and oil and, therefore, a number of hydroelectric plants have and are being installed in India to harness the available water power. Turbines are designed for specified conditions of head, speed and output. However, they may actually be required to operate under conditions quite different from designed point. It is therefore, desirable to have complete information about the performance characteristics of turbine under all possible operating conditions, and make it available to the user. Experiments are, therefore, conducted on the manufactured turbine, in the manufacturer's laboratory before it is supplied to the user. In case the turbine is too big for laboratory experiments, a small scale model of reasonable dimensions is prepared and tests are conducted the same prior to fabrication of the prototype. The experimental results obtained from the model turbine are plotted

and interpreted for the performance of the actual turbine. The performance curves are known as the "characteristic curves" of the turbine.

BACKGROUND INFORMATION:

A pelton turbine is an impulse or free jet turbine. In such a turbine the available hydraulic energy is converted into kinetic energy by means of an efficient nozzle. The jet issuing from the nozzle strikes a series of suitably shaped buckets fixed around the rim of a wheel and forcing it to rotate. The turbine operates under atmospheric pressure. In an experimental system the water is usually supplied to the turbine by means of a centrifugal pump. The rate of flow is measured by means of a calibrated orifice plate. The discharge to the turbine can be varied by changing the nozzle opening by means of a spear valve. The head of the turbine can be varied by adjusting a valve installed in the delivery pipe of the pump. Alternately the head can be varied by changing the speed of the motor driving pump, if it is a D.C. motor. The speed of the turbine is measured by means of a tachometer and the output by means of a prony brake.

PROCEDURE :

The student should examine their experimental rig and work out the procedure to be adopted to fulfil the objective of the experiment. They should be in a position to ask the laboratory technician for the accessories and instruments necessary to do the work. It may be noted that there is a necessity for priming of the supply pump. Also there is a need to note various initial readings and to plan the steps for taking observations. The student should plan which performance curves are to be drawn. For example, one possibility is to study the performance of the Pelton turbine at a constant head and a constant spear opening; alternately, the performance may be studied at a constant head and constant speed. For the former set of characteristics the desired output may be curves of efficiency, discharge rate and unit power, each versus the unit speed. The experimental planning is, therefore, done in consonance with the desired objectives.

RESULTS

1. Calculate the head on the turbine (H) from the expression equation $H = H_m + Z + \frac{U_p^2}{2g}$ equation Where H_m is the pressure gauge reading (m), U_p is velocity of flow in the pipe (m/s) at the cross-section where the pressure gauge is connected and Z is the datum head of the pressure gauge above the centre-line of the nozzle (m).
2. Calculate the water power from the expression equation $P_i = \rho g Q H$ equation Where ρ is the mass density of water (kg/m^3) and Q is the discharge rate through the turbine (m^3/s). Calculate the brake power from the expression equation $P_o = T\omega$ equation where T is the torque measured by the brake system in N-m and ω is the speed of the turbine in radians per second.
3. Calculate the overall efficiency of the turbine from the expression $\eta = \frac{\text{Brake power}}{\text{Water power}}$ equation
4. Calculate the unit quantities from the basic measurements
 - (a) Unit speed $= \frac{N}{\sqrt{H}} = N_u$
 - (b) Unit discharge $Q_u = \frac{Q}{\sqrt{H}}$
 - (c) Unit power $P_u = \frac{P_o}{H^{3/2}}$

where N is rotation rate in RPM.

5. Plot the results at each nozzle opening (or spear valve setting) in the form of the graphs:
 - (a) η_0 vs N_u
 - (b) Q_u vs N_u
 - (c) P_u vs N_u

POINTS FOR DISCUSSION :

- (a) Describe the results of the experiment in terms of the trends discernible from the characteristic graphs.

- (b) Compare the experimental results with the expected trends reported in literature.
- (c) Define the term run-away speed as applied to a turbine. What is its significance?
- (d) Find the dimensionless specific speed of the experimental turbine. Is it within the range of specific speeds for the normal operation of a Pelton wheel? operation of Compare experimental results with the values given in standard texts.
- (e) What type of arrangement would be necessary for the production of large power by means of a Pelton turbine.
- (f) What measures are needed to improve the efficiency of the turbine?
- (g) Compare the trends of performance of a Pelton wheel with the performance of a Francis turbine. Record the relative merits of one vis-a-vis the other regards their performance.
- (h) Would the installation of a draft tube be helpful in improving the head or efficiency of a Pelton turbine?
- (i) Would characteristic plot on the basis of specific quantities be as good as those on the basis of unit quantities?
- (j) How can the performance of a Pelton turbine whose length dimensions are scaled up by a factor of n in comparison with the experimental turbine be predicted from the results of present experiment?
- (k) How is the experiment on "the impact of jet on vanes" related to the study of a Pelton turbine? Comment on the value of that experiment where vane are fixed in the contrast to that for a Pelton wheel where the vanes must move.

Experiment 3

Conservation of Mechanical Energy (Bernoulli's Theorem)

Objective

To evaluate the different components of mechanical energy possessed by a fluid element at different sections of a converging-diverging duct and thus appreciate the limitations of Bernoulli's equation applied to incompressible flows.

Practical Relevance

In our daily lives we consume a lot of water for various reasons. This water is delivered through taps connected to a network of pipes of different sizes, which in turn receives water from an overhead tank or a pump. How does one estimate pumping losses or calculate the flow at various points? One of the equations used in such engineering calculations is Bernoulli's equation (principle of conservation of mechanical energy). The present experiment illustrates this principle as applied to simple practical flow systems and also highlights its limitations.

Theoretical Background

The law of conservation of energy as applied to fluid systems is essentially the first law of Thermodynamics. If we consider mechanical energy only, we have three contributions, namely:

- potential energy
- kinetic energy
- pressure energy

If we make some simplifying assumptions, then the expression for conservation of mechanical energy per unit volume becomes,

$$p + \frac{1}{2}\rho V^2 + \rho g z = \text{constant} = \text{total energy per unit volume} \quad (3.1)$$

Here, p is pressure, ρ is density, V is fluid velocity, and g is acceleration due to gravity.

In engineering applications Eq. (3.1) is usually expressed in terms of head or energy per unit weight and then this form of Bernoulli's equation is,

$$\frac{p}{\rho g} + \frac{V^2}{2g} + z = h + \frac{V^2}{2g} = \text{constant} = H \quad (3.2)$$

Here, h is piezometric head and H is total head.

The assumptions are that the fluid is ideal ($\mu = 0$), it is incompressible ($\rho = \text{constant}$) and that the flow is irrotational, steady and uniform. In a real flow, these assumptions are violated and some energy loss does occur and hence Bernoulli's equation is strictly not valid. This experiment helps highlight these limitations.

Experimental set-up

The rig consists of a constant-head supply tank connected to a converging-diverging duct with piezometric tubes, a volumetric tank at the outlet, a sump and a pump. A schematic diagram of the experimental setup is shown in figure 3.1.

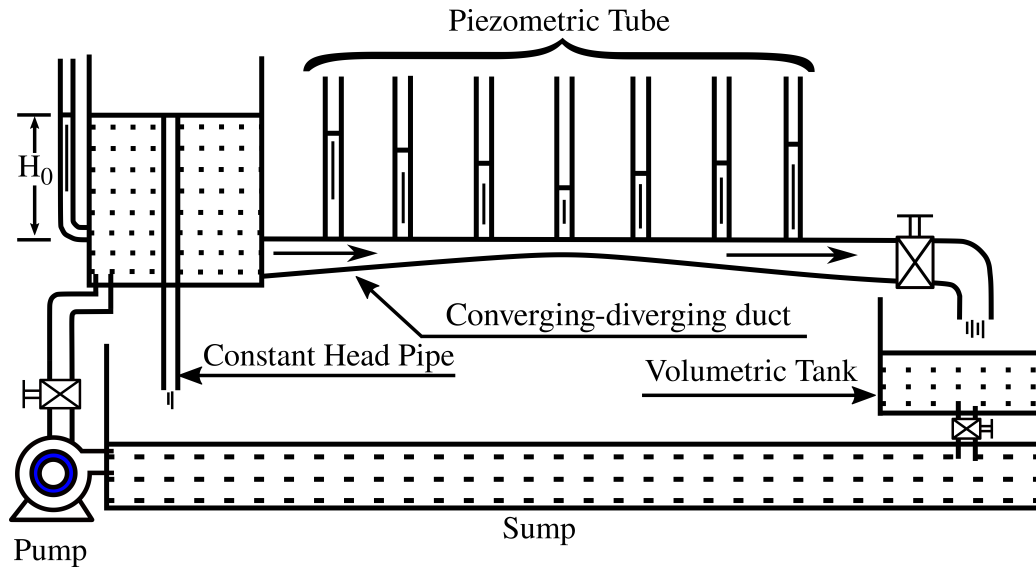


Figure 3.1: Schematic diagram of the experimental setup

Procedure

- Measure the location of each piezometric tube with respect to the supply tank outlet and the area of cross-section of the converging-diverging duct at these locations (if these are not given).
- Adjust the flow rate control valve at the exit of the converging-diverging duct to any setting and also adjust the constant head pipe to a reasonable height. Next, adjust the flow rate of the pump so that the overflow from the constant head pipe is quite small and there are no surges in the tank.
- Measure the supply tank head, H_0 .
- Measure the piezometric head in each tube.
- Determine the flow rate using either the volumetric or gravimetric methods.
- Repeat steps 2 through 5 for one more flow rate, obtained either by changing the supply tank head or the valve setting at the exit of the converging-diverging duct.

Observations

- (a) Cross-sectional area of the volumetric tank, $A =$ m^2

Flow rate 1

- (b) Supply tank head, $H_0 =$ m
- (c) Time taken for $0.1m$ rise in the volumetric tank level, $\Delta t(s)$
 1) 2) 3)
 Average $\Delta t =$ s .
- (d) Volume flow rate, $Q = 0.1A/\Delta t =$ m^3/s
- (e) Observation Table

Table No.	Distance from the tank, $x(m)$	Cross-sectional area, $a(m^2)$	Piezometric Head, $h(m)$
1			
2			
3			
4			
5			
6			
7			
8			
9			
10			
11			
12			
13			
14			
15			

Table 3.1: Observation table

- (f) Data analysis

Tube No.	Distance	Piezometric Head, h (m)	Flow Velocity $V = Q/a$ m/s	Kinetic Energy Head (m)	Total Head, H (m)	Loss Coefficient, $C = (H_0 - H)/H_0$

Table 3.2: Analysis table

(g) Plot the variation of:

- i. Piezometric head along the duct.
- ii. Kinetic head along the duct.
- iii. Total energy head along the duct.
- iv. The loss coefficient along the duct.

(Note: plotting the first three plots together is recommended.)

(h) Repeat steps 2 through 6 for the other flow rate.

Discussion

Highlight what you consider to be the key results of the experiment and discuss the results with reference to the above plots.

List out the source of errors and suggest methods for improving the experiment.

Experiment 4

Study of the Mean Flow of a Free Air (Submerged) Jet

Objective

To study the mean flow characteristics of a free air jet and to study the application of the integral forms of the continuity, momentum and energy equations.

Practical Relevance

The term submerged jet refers to a liquid jet flowing in another miscible liquid or to a gaseous jet flowing in air. Laminar and turbulent submerged jets find wide engineering application. In many combustors, gaseous fuel is introduced in the form of a high velocity jet into a co-flowing air stream. The plume emanating from a factory chimney, the exhaust of a jet engine, industrial waste efflux into rivers and seas, are other examples of submerged jets.

Theoretical Background

When a jet exits from a well designed nozzle, as shown in figure 1, the velocity profile at the exit is nearly uniform. The development of such a jet can be described in terms of three distinct zones:

- (a) the potential core zone.
- (b) the zone of establishment.
- (c) the fully developed zone.

As the jet interacts with the surrounding air (or liquid), the velocity at the periphery begins to decrease due to mixing with the surrounding still air, however, initially the velocity in the central region of the jet is the same as the exit velocity. As the jet develops this central region, called the potential core, begins to diminish in size and at the start of the zone of establishment the potential core disappears and the centre-line velocity drops below the exit velocity. In the zone of establishment the mean velocity profile gradually changes shape till it becomes nearly Gaussian and in the fully developed zone, although the jet continues to spread linearly, and the centreline velocity continues to decrease, the shape of the mean velocity profile does not change. If the exit area of the jet is A , and its exit velocity is U_o , then the initial volume flux, Q_0 : the initial momentum flux M_0 , (per unit volume) and the initial energy flux E_0 (per unit volume), are given by the following:

$$Q_0 = U_o A; M_0 = AU_o^2; E_0 = AU_o^3 \quad (4.1)$$

At any other downstream location, for an axisymmetric jet, if $u(r)$ denotes the downstream velocity as a function of the radius, r , the volume flux, Q , the momentum flux, M , and the energy flux, E , are given by,

$$Q = \int_0^\infty u 2\pi r dr, \quad (4.2)$$

$$M = \int_0^\infty \rho u^2 2\pi r dr, \quad (4.3)$$

$$E = \int_0^\infty \rho u^3 2\pi r dr. \quad (4.4)$$

Since there is no external force acting on the jet after it leaves the nozzle, it is quite easy to show that for any axial position, the momentum flux is constant and is equal to M_0 . In reality one cannot integrate up to $r = \infty$ and thus the integration is carried out up to $r = R$, where the velocity, u , is nearly zero. Thus M is approximately constant.

In the experiment the velocity is measured with the help of a pitot tube connected to an inclined manometer, making an angle, θ , to the horizontal. If ρ_m is the density of the manometric fluid then,

$$u(r) = \sqrt{2\left(\frac{\rho_m}{\rho} - 1\right)gh(r) \sin \theta} \quad (4.5)$$

where h is the manometer deflection.

Since $\rho_m \gg \rho$ we can simplify equation (9.5) as follows,

$$u(r) = \sqrt{2\frac{\rho_m}{\rho} gh(r) \sin \theta} \quad (4.6)$$

Experimental set-up

The rig consists of a blower which is connected to a settling chamber fitted with a nozzle at the exit. The measurements are carried out with a pitot tube that can be positioned at various points in the flow by means of the traverse mechanism shown. The output of the pitot tube is connected to a U- tube manometer with one end open to atmosphere.

Procedure

- (a) Measure the ambient temperature and atmospheric pressure. This will enable you to get the correct value for the density of air, ρ , used in the calculation of velocities.
- (b) Note down the nozzle diameter, d , and the horizontal scale factor.
- (c) With the help of the traverse mechanism, lower the pitot tube such that its tip is at an axial distance, $x/d = 0.5$. Centre the pitot tube with the help of the plumb-bob provided, and reset the zero on the horizontal scale.

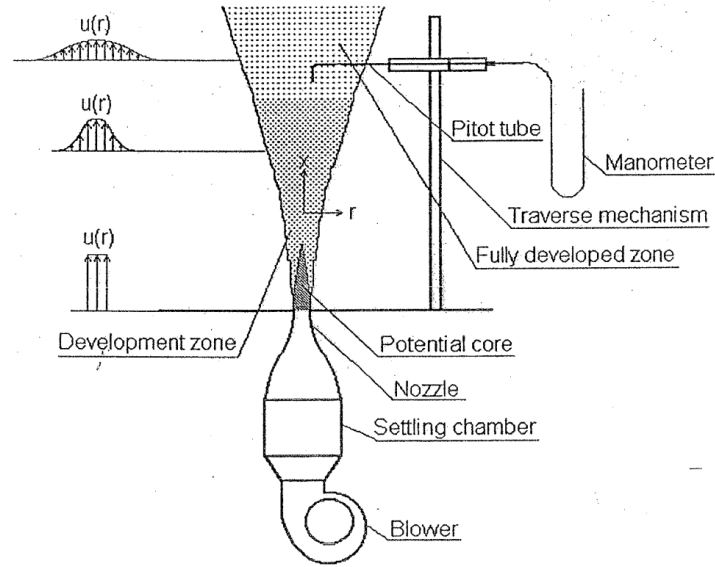


Figure 4.1: Schematic diagram of the experimental setup

- (d) Move the pitot tube away from the centre slowly while keeping an eye on the manometer deflection. Note the scale reading, n , when the deflection goes to zero. Between the centre (0) and n take 8 velocity readings to get the velocity profile.
- (e) Repeat steps c and d for four other locations, viz., $x/d = 4, 8, 12$ and 16.
- (f) For each x/d , do the following:
 - i. Plot $u(r)$ v/s r .
 - ii. Fit a smooth curve through the data points and from the curve read off values of $u(r)$ at equal intervals, approximately 30 points in all.
 - iii. Prepare tables of $ru(r)$, $ru^2(r)$, $ru^3(r)$, for different r and integrate numerically to obtain Q, M and E .
- (g) Plot Q/Q_0 , M/M_0 , and E/E_0 , against x/d , all on the same graph sheet.

Observations

- (a) Room Temperature=... °C; Atmospheric Pressure =... mm Hg
- (b) Conversion factor for the traverse mechanism =... units/mm
- (c) Angle of inclination of the manometer, $\theta = \tan^{-1}()$ = ...

LOCATION 1

- (d) x/d =
- (e) Scale reading for which the manometer deflection is zero = ...

Observations table

Serial No.	Scale Units	Radius,r,mm	Manometer Reading,h,mm
1			
2			
3			
4			
5			
6			
7			
8			

Repeat steps d) and e) for the other axial positions.

Discussions

Highlight what you consider to be the key results of the experiment and discuss the results with reference to the above graph.

Error Analysis

List out the sources of errors and do a detailed error analysis for all the measurements. Estimate the expected error in the final result. Suggest methods for improving the experiment.

Experiment 5

Performance Characteristics of Free Jets

Objective

1. To evaluate the performance parameters, namely, discharge coefficient, velocity coefficient and contraction coefficient for a free jet flowing through an orifice or a nozzle.
2. To appreciate the limitations of Torricelli's Theorem which states that the velocity of efflux from a tank open to the atmosphere is the same as that of a rigid body falling from a height equal to the level in the tank above the vent.
3. ~~To determine the time of emptying of the tank.~~

Practical Relevance

Small openings are often provided in storage tanks to provide a means of quickly emptying them by discharging the enclosed fluid as a free jet into the atmosphere. A knowledge of the performance of free jets is also useful in the design of overhead sprinklers for fire safety. ~~On the other hand, if a tank storing hazardous material were to develop a leak, it would be very important to know how long it would take for a sizeable quantity to drain out and how quickly it should be contained.~~

Theoretical Background

Consider the flow taking place from a round opening (orifice or nozzle) provided in the side of a large tank as shown in Figure 5.1 below. If one assumes that the area of the tank is much larger than that of the opening, then, applying Bernoulli's equation between the surface and the exit (points (1) and (2) respectively), we have,

$$\frac{p_{atm}}{\rho g} + H = \frac{p_{atm}}{\rho g} + \frac{V^2}{2g} \quad (5.1)$$

Thus,

$$V = \sqrt{2gH} \quad (5.2)$$

By applying Bernoulli's equation we are tacitly assuming that there are no losses between points (1) and (2). In reality the exit velocity, V , is less than that given by Equation 5.2 and this loss is characterised by the velocity coefficient, C_v , which is the ratio of the actual velocity at the exit to the theoretical estimate given in Equation 5.2. If the exit is well rounded the losses are minimum and C_v is close to 1.

Another effect comes into play if the opening is not well rounded. The flow in the tank occurs over a much larger area than the opening, and thus streamlines are converging as they approach the exit. If the exit is not provided with a long straight section, the streamlines will continue to converge after the exit, thus reducing the effective area of the opening. The minimum area thus formed is called the vena-contracta. The ratio of the area at the vena-contracta, A_{vc} , to the area of the opening, A , is called the coefficient of contraction, C_c . The value of C_c varies from 0.6 to 1 depending on the type of opening.

The rate of efflux from the opening may now be estimated as,

$$Q = C_c C_v \sqrt{2gH} A \quad (5.3)$$

We can define a discharge coefficient, C_d , as the ratio of the actual discharge to the theoretically expected discharge. Thus,

$$C_d = \left(C_c C_v \sqrt{2gH} A \right) / \left(\sqrt{2gH} A \right) = C_c C_v \quad (5.4)$$

The value of C_d may be estimated experimentally in two ways. The first method is to directly determine the actual flow rate for a given head, H ,

by either the volumetric or the gravimetric method and then calculate C_d . The second method is to let the tank drain from an initial height H_1 to a final height H_2 and find the elapsed time Δt . Then C_d may be estimated as follows: (Note that the total drop in height should be small compared to H_1 for this method to be accurate.)

$$Q = -A_t \frac{dH}{dt} \quad \left[\frac{dH}{dt} \text{ is negative} \right] \quad (5.5)$$

Where, A_t is the cross-sectional area of the tank.

Also,

$$Q = C_d A \sqrt{2gH} \quad (5.6)$$

Thus,

$$C_d dt = - \left(\frac{A_t}{A} \right) (2gH)^{-\frac{1}{2}} dH \quad (5.7)$$

$$C_d = 2 \left(\frac{A_t}{A} \right) (2g)^{-\frac{1}{2}} \left(\sqrt{H_1} - \sqrt{H_2} \right) / \Delta t \quad (5.8)$$

The two estimates of C_d may be compared and Equation 5.4 can also be verified in the experiment.

Experimental set-up

The rig consists of a constant-head supply tank with an opening on the side (orifice or nozzle), a volumetric tank at the outlet, a sump and a pump. A schematic diagram of the set-up is shown in Figure 5.1.

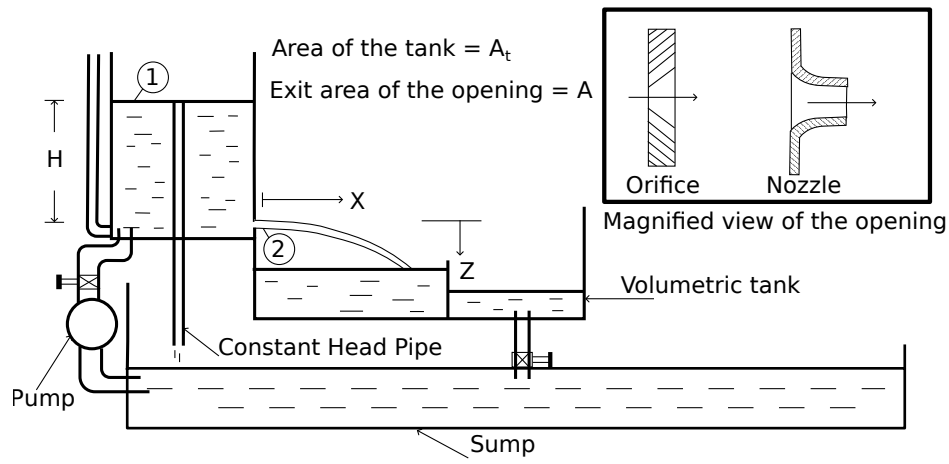


Figure 5.1: Schematic diagram of the set-up

Procedure

1. Measure the outlet diameter of the device to be tested (orifice or nozzle) and fit it on to the side of the tank.
2. Start the pump and then open the supply line to the tank. Stop up the opening so that the tank fills faster. Release the opening after water starts to overflow from the constant head pipe and then adjust the flow rate such that the overflow is very small so as to prevent surges in the tank.
3. Measure the constant head in the supply tank, H .
4. Determine the flow rate using either the volumetric or the gravimetric method.
5. Measure the diameter of the vena-contracta using the Vernier callipers provided.
6. Measure the contour of the free jet with the help of the travelling scale provided. Take $x = 0$ and $z = 0$ at the vena-contracta. (Measure the upper surface of the jet only.)
7. Repeat steps 1 through 6 for the other type of opening.
8. The second part of the experiment concerns the time of emptying of the tank. Choose the nozzle for this part. Close the valve in the supply line and shut off the pump.

9. ~~Measure the time of emptying for every 5cm drop in the water level in the tank, till it is nearly empty (only about 5cm left). (Note that only the first 3 to 4 readings may be used to estimate C_d .) Also note any qualitative changes in the jet trajectory.~~

Observations:

- a) Cross-sectional area of the volumetric tank, $a = \text{---} \text{m}^2$
 b) Cross-sectional area of the supply tank, $A_t = \text{---} \text{m}^2$

Readings for the Nozzle / Orifice:

- c) Supply tank head, $H = \text{---} \text{m}$
 d) Diameter of the opening, $D = \text{---} \text{m}$
 e) Time taken for 0.1m rise in the volumetric tank level, Δt (s):
 1) ----
 2) ----
 3) ----
 Average $\Delta t = \text{---} \text{s}$
 f) Volume flow rate, $Q = 0.1 a / \Delta t = \text{---} \text{m}^3/\text{s}$
 g) Diameter of the vena-contracta, $D_{vc} = \text{---} \text{m}$
 h) Contraction coefficient, $C_c = (D_{vc}/D)^2 = \text{---}$
 i) Observation Table 1:

Sr. No.	Horz. Distance, x (m)	Vert. Distance, z (m)	$z_{th} = x^2/(4H)$ (m)	$C_v = x/(2\sqrt{zH})$
1				
2				
3				
4				
5				
6				
7				
8				
9				

Table 5.1: Observation Table 1

Average $C_v = \text{----}$

Repeat steps c) through i) for the other type of opening.

j) Observation Table 2 (For the nozzle only)

Serial Number	1	2	3	4	5	6
Level of water in the tank, H_i (m)						
Time elapsed, t_1 (s)						
Theoretical estimate ($C_d = 1$), t_2 (s)						
$C_d = t_2/t_1$						

Table 5.2: Time of Emptying - Part 1

Serial Number	7	8	9	10	11	12
Level of water in the tank, H_i (m)						
Time elapsed, t_1 (s)						
Theoretical estimate ($C_d = 1$), t_2 (s)						
$C_d = t_2/t_1$						

Table 5.3: Time of Emptying - Part 2

Average $C_d = \text{----}$

k) Plot the following

- i) Theoretical and experimental trajectories for the two devices.
- ii) ~~Theoretical and experimental time of emptying versus the level in the tank.~~

Discussions:

- Highlight what you consider to be the key results of the experiment and discuss the results with reference to the above graphs.
- List out the sources of errors and suggest methods for improving the experiment.

Experiment 6

Obstruction Type Flow Meters

Objective

To understand the behaviour of various obstruction type flow meters, namely the venturi-meter, the nozzle and the orifice plate and to calibrate them over a range of flow rates.

Practical Relevance

Quick and reliable flow rate measurement is of utmost importance in any industry handling fluids. Almost always one needs to measure the flow rate within ducts or confined flows thus the volumetric or gravimetric method is out of question. Obstruction meters of the type listed above are the most commonly used devices. The actual meter used depends on the fluid being handled, the accuracy required and the energy loss that can be tolerated.

Theoretical Background

For any incompressible fluid, if the flow is forced to go through a contraction, the velocity has to increase in for mass balance. This increase in velocity causes the pressure to drop locally (Bernoulli's equation).

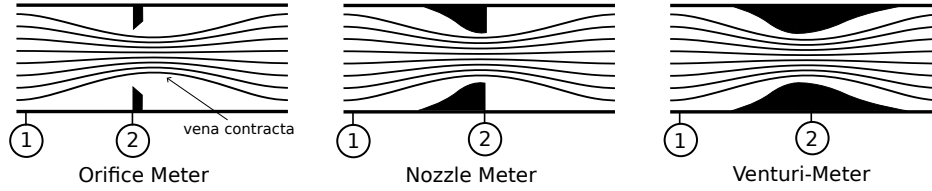


Figure 6.1: Flow meters

For well designed nozzles and venturis the minimum flow area occurs at the minimum area of the device as shown. However, due to the abrupt contraction in the case of the orifice meter, the streamlines continue to converge after the device — the minimum area thus formed is called the *vena contracta*. If we ignore this extra contraction and apply Bernoulli's equation between points 1 and 2 for each of the devices in Figure 6.1, we have,

$$Q_{th} = \frac{A_1 A_2 \sqrt{2g(1 - \rho_m/\rho_f)h}}{\sqrt{A_1^2 - A_2^2}} \quad (6.1)$$

Here, Q_{th} is the volume flow rate, A_1 and A_2 are the areas at sections 1 and 2 respectively, ρ_f is the density of the working fluid (water), ρ_m is the density of the manometric fluid (air) and, h , is the difference in the levels of the two limbs of the manometer, connected between sections 1 and 2.

In actual practice, there are energy losses in the region between sections 1 and 2, thus the actual flow rate Q_{act} is less than Q_{th} . The expression for Q_{act} may then be written in terms of a discharge coefficient, C_d , as follows:

$$Q_{act} = C_d \frac{A_1 A_2 \sqrt{2g(1 - \rho_m/\rho_f)h}}{\sqrt{A_1^2 - A_2^2}} \quad (6.2)$$

The value of C_d depends on the type of device and also on the exact location of the pressure taps.

Experimental set-up:

The rig consists of a constant-head supply tank connected to three supply lines (one for each device to be tested) by means of a common manifold, a volumetric tank at the common outlet, a sump and a pump. A schematic diagram is shown in Figure 6.2.

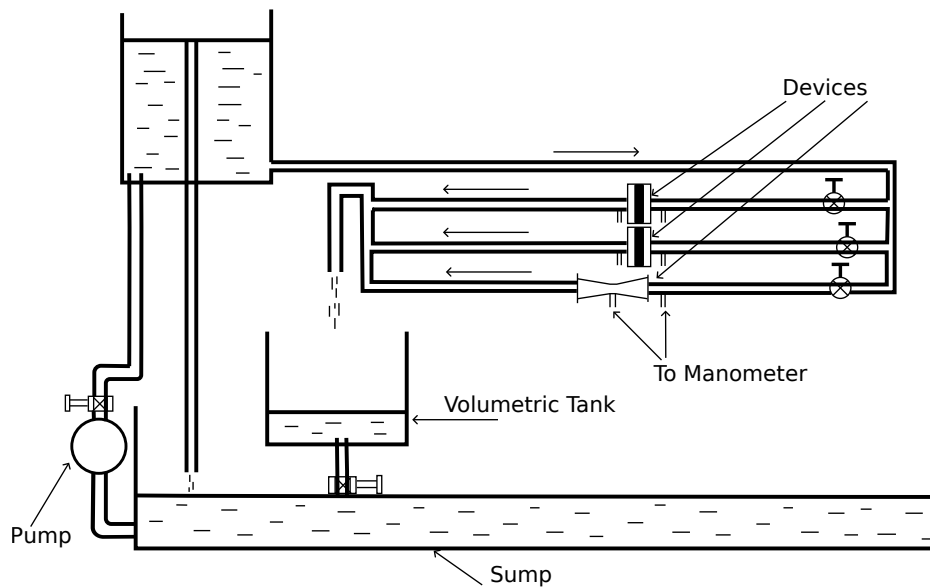


Figure 6.2: Schematic diagram of the set-up

Procedure

1. Measure (note down) the pipe diameters and A_1 and A_2 for the three devices.
2. Start the pump and then adjust the constant head pipe to achieve the desired head.
3. Choose one of the three devices and open completely the valve on the input line to this device while keeping the other two valves closed.
4. Measure the pressure differential across the device using the inverted U-tube air manometer provided.
5. Determine the flow rate using either the volumetric or the gravimetric method.
6. Repeat steps 5 and 6 for five more flow rates, progressively reducing the flow rate such that the pressure differential is approximately 1 cm of water for the lowest flow rate.
7. Repeat steps 4 through 7 for the other two devices.

Observations

a) Cross-sectional area of the volumetric tank, $a = \quad \text{m}^2$

DEVICE 1:

b) Pipe diameter, $D_1 = \quad \text{m}$

c) Throat diameter, $D_2 = \quad \text{m}$

d) $A_1 = \quad \text{m}^2$, $A_2 = \quad \text{m}^2$

e) Table 1

Sr. No.	Manometer Reading h (water), m	Height filled in Volumetric Tank h' , m	Time taken for Filling t , s
1			
2			
3			
4			
5			
6			

Table 6.1: Observation Table

f) Table 2

Sr. No.	Actual Flow rate $Q = ah'/t \text{ m}^3/\text{s}$	Theoretical Flow rate Q_{th} (equation 1) m^3/s	Discharge Coefficient C_d	Reynolds Number Re
1				
2				
3				
4				
5				
6				

Table 6.2: Analysis Table

(Here the Reynolds Number Re is defined as $QD_1/(A_1\nu)$)

h) Repeat steps c) through f) for the other two devices.

g) Plot the following (for each of the devices):

i) Q v.s. $\sqrt{h}$

ii) C_d v.s. Re

Discussions

- Highlight what you consider to be the key results of the experiment and discuss the results with reference to the above graphs.
- List out the sources of errors and suggest methods for improving the experiment.

Experiment 7

Performance Characteristics of a Centrifugal Pump

OBJECTIVE:

To study the performance characteristics of a single stage centrifugal pump.

BACKGROUND INFORMATION

Centrifugal pumps are rotodynamic machines. It develops dynamic pressure and lifts liquid from lower level to a higher level. The main component of a centrifugal pump is an impeller (a wheel or a rotor) provided with a series of backward curved vanes or blades. The other components are a casing (air-tight chamber) surrounding the impeller and a bracket comprising stuffing box and bearings. The pump is always installed with a suction pipe connected to the centre or the eye of the impeller with its lower end dipped into the liquid being sucked and a delivery pipe connected to the outlet of the pump. The impeller is mounted on a shaft which is coupled to an electric motor or a diesel engine that imparts energy to the impeller and causes it to rotate. The centrifugal pump is initially primed wherein the suction pipe, casing of the pump and the portion of the delivery pipe upto delivery valve are completely filled with the liquid to be pumped. With the delivery valve closed, the impeller is made to rotate. As a result a forced vortex is developed which imparts a centrifugal head to the liquid. Opening the delivery valve forces the liquid to flow in an outward radial direction thereby leaving the vanes

of the impeller at the outer circumference with high velocity and pressure. The high pressure of liquid leaving the impeller is continuously supplied with replacement liquid from the sump as a result of the suction pressure in the suction pipe (a partial vacuum exists at the eye of the impeller and liquid in the sump is at atmospheric pressure). The high absolute velocity at the outlet of the vanes is converted to useful pressure energy by shaping the casing such that the liquid flows through a gradually expanding passage. In summary, it may be stated that a centrifugal pump lifts the liquid to a higher level as a result of modification of the hydraulic gradient caused by centrifugal action and change in angular momentum. This is in contrast to a positive displacement pump wherein lifting is due to pushing in a confined space.

The main advantages of centrifugal pump vis-a-vis positive displacement pump is that its discharge capacity is much greater, it can be used to pump highly viscous liquid also, it can be operated at high speeds with less danger of separation and cavitation, and its maintenance requirements are low. However, it can not built-up pressures as high as those that can be built-up by reciprocating pumps.

While studying the characteristics of a centrifugal pump, its speed is kept constant throughout. The head, H , the input power P_i , and the output power, P_o , are then determined as a function of volume flow rate Q .

The head H is defined as the total effective head generated by the pump and is given by the expression :

$$H = \left(H_{gd} + Z_d + \frac{U_d^2}{2g} \right) - \left(H_{gs} + Z_s + \frac{U_s^2}{2g} \right)$$

Where H_{ad} and H_{gs} are the delivery and suction gauge readings in meters of water, respectively, Z_d and Z_s , are the vertical distances of the delivery and suction gauges from the pump axis, respectively, and $U_d^2/2g$ and $U_s^2/2g$ are the available kinetic energy per unit weight of the pumped liquid in the delivery and suction pipe respectively, at the sections where the gauges are fitted. The input power P_i , is the motor power determined from the dynamometer or otherwise. The output power P_o , is the water power determined from the expression $P_o = \rho g Q H$ where ρ is the mass density of liquid. The overall efficiency of the pump is then given by

$$\eta = \frac{P_o}{P_i}$$

The performance curves indicate the behaviour of the particular centrifugal pump under different operating conditions. For large centrifugal pumping units, the performance characteristics of dynamically similar model is initially determined and then the theory of similitude is used to predict the performance characteristics of the prototype. Affinity laws are applied to determine the performance of similar (homologous) pumps pumping same liquid and are as follows:

$$\begin{aligned}\frac{Q}{ND^3} &= \text{Constant} \\ \frac{H}{N^2D^2} &= \text{Constant} \\ \frac{P}{N^3D^3} &= \text{Constant}\end{aligned}$$

Where

- D = outlet diameter of the impeller
- N is pump rotational speed in rad/s
- P = input power
- Q is the discharge rate in m^3/s
- H is the head developed in meters of liquid column

Homologous pumps have same specific speed, which is represented as,

$$N_s = \frac{N\sqrt{Q}}{(gH)^{3/4}}$$

where g gravitational constant in m/s^2

PROCEDURE:

The students are advised to check the experimental rig existing in the laboratory and to plan their experiment themselves. Given the objectives and some background information on centrifugal pumps, they should ask for what they want and check out the steps of the procedure and the calculations. Care should be taken to take all the initial readings and to start and stop the pump with due care. They can, then record their procedure details and state why they choose so in the particular manner chosen.

RESULTS:

The following graphs may be drawn for operating at constant speed:

1. Total head generated H vs discharge rate Q .
2. Power input P , vs discharge rate Q .
3. Overall efficiency η vs discharge rate Q .

The students may, of course, prefer to draw the curves on non-dimensional basis and interpret their curves.

POINTS FOR DISCUSSION:

1. Discuss the nature of the curves plotted and comment on the expected shapes of the curves if the experiment is performed at different speeds.
2. Can cavitation take place in the pump? If so, where and why ?
3. Calculate the specific speed of the pump and comment on its value.

Experiment 8

Flow Visualization

Objective

~~To use two simple flow visualization techniques and to study streamline patterns around various objects placed in a fluid stream.~~

To study streamline patterns formed around various objects placed in a fluid stream using a fundamental flow visualization technique.

Practical Relevance

In the study of complex flows, the first step usually taken is to visualize it in some manner. This way, one obtains a qualitative picture before rigorous quantitative analysis is begun. No design of hydrodynamic or aerodynamic machines is complete without a detailed qualitative analysis of flow. Theoretical Background: In the study of flow patterns, three different descriptors may be used.

- Streamline: This is an imaginary line drawn through the flow field, at any instant, such that everywhere its tangent points in the direction of the local velocity vector.
- Pathline: As the name suggests this is the line traced out by a fluid particle over a period of time.
- Streakline: This is the locus, at any instant, of all particles that have passed through a fixed point in space, at some previous time.

It is easy to see that for steady flow, all three lines described above are identical. Mathematically, it is the streamline that is most useful; however, it is almost impossible to generate it directly by experiments. On the other hand, the streakline is most difficult conceptually, but it is the easiest to generate in the laboratory, for which one simply introduces dye continuously at a fixed location and watches the streak develop after some time.

In the experiment, the flow is maintained nearly steady so that the observed streakline patterns are approximately the same as the expected streamline patterns.

There are many techniques used for flow visualization:

- Dye injection and powder sprinkling - (used in liquid flows)
- Smoke wire or smoke tube - (used for gaseous flows)
- Wool tuft method - (used for gaseous flows)
- Hydrogen bubble method - (used in water)
- Schlieren technique - (used in compressible flows), etc.

Here we will be studying the dye injection and powder sprinkling techniques.

Experimental set-up

The rig consists of a water table that is made very flat and as horizontal as possible. There is an inlet tank and an outlet tank on either side of this water table. The flow of water is regulated with a control valve connected at the pump exit. A schematic diagram of the set-up is shown in figure 8.1.

Procedure

Dye injection and powder sprinkling method

1. Place the cylinder on the water table and start the pump. Ensure that the flow on the table is continuous and without any dry patches.
2. Sprinkle some Potassium Permanganate (KMnO_4) crystals upstream of the cylinder and adjust the flow speed such that the observed pattern is nearly steady. (You need a fairly low speed to achieve this.)
3. By judiciously placing the KMnO_4 crystals, observe the overall streak-line pattern.

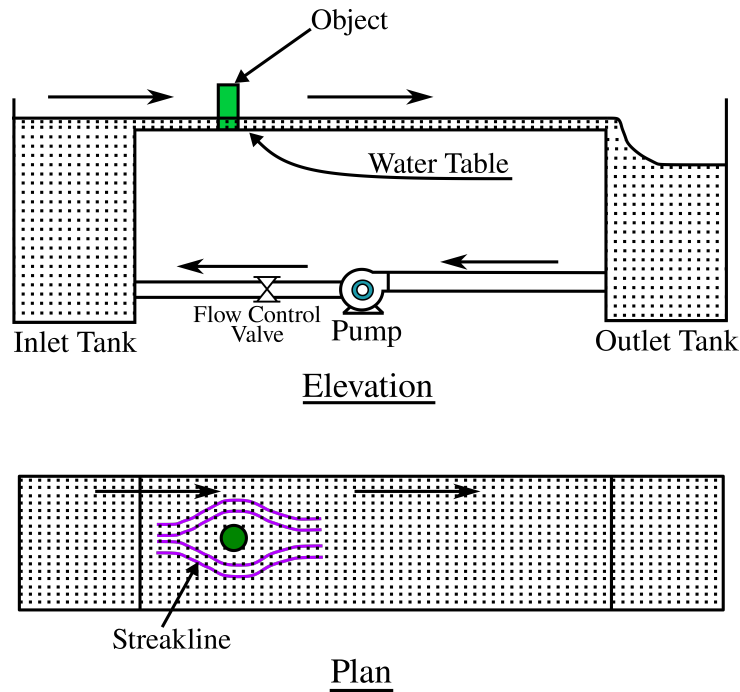


Figure 8.1: Schematic diagram of the experimental setup

4. Sprinkle Lycopodium powder upstream and downstream of the cylinder and observe flow features like stagnation points, separation points, recirculation or dead zones etc.
5. Make sketches of the flow patterns for inclusion in your report.
6. Measure the surface flow speed by measuring the time taken for a small patch of Lycopodium powder to travel a distance of $1m$ (in the absence of the cylinder).
7. Keeping the flow rate constant, repeat steps 3, 4 and 5 for the other objects provided: the aerofoil (at various angles of attack), rectangular body (oriented both ways to the flow), oval cylinder (oriented both ways to the flow), T-section, bi-convex airfoil, etc.
8. Observe the starting vortex for the aerofoil kept at a non-zero angle of attack to the flow.
9. Measure the projected length perpendicular to the flow of all the objects used in the steps above.

- Place the cylinder in the flow again and increase the flow speed in small steps till the flow becomes fully turbulent. Describe the intermediate stages

Smoke method

- Observe the flow pattern around various objects placed inside the smoke tunnel and make sketches of the same.

Observations

- Time taken for the Lycopodium powder to travel $1m$, Δt (s):
 a)..... b)..... c).....
 Average $\Delta t_{average} =$ s
- Velocity, $U_1 =$ m/s
- Kinematic Viscosity of water, $\nu =$ m^2/s
- Time taken for the Lycopodium powder to travel $1m$, for increased flow rate Δt (s):
 a)..... b)..... c).....
 Average $\Delta t_{average} =$ s
- Velocity, $U_2 =$ m/s
- Time taken for the Lycopodium powder to travel $1m$, for increased flow rate Δt (s):
 a)..... b)..... c).....
 Average $\Delta t_{average} =$ s
- Velocity, $U_3 =$ m/s
- Sketch the various flow patterns.
- Observation table:

(The Reynolds number is defined as, $R = UL/\nu$)

Discussion

- Write short notes (3 – 4 lines) on each of the observed flow patterns in relation to the calculated Reynolds number.
- What are the challenges encountered in the experiment, and what steps may be taken to make them better?

Object Name and Orientation	Flow velocity, U (m/s)	Projected length, L (m)	Reynolds Number, R
Cylinder (For V_1)			
Rectangle (narrow side along the flow)			
Rectangle (broad side along the flow)			
Airfoil (low angle of attack)			
Airfoil (high angle of attack)			
Oval (narrow side along the flow)			
Oval (broad side along the flow)			
Biconvex airfoil (zero angle of attack)			
Biconvex airfoil (high angle of attack)			
Cylinder (For V_2)			
Cylinder (For V_3)			

Table 8.1: Observation table

Additional readings

Experiment 9

Flow Around a Circular Cylinder

Objective

To determine the drag force on a circular cylinder placed in a uniform stream of air.

Practical Relevance

Flow around cylindrical (circular and non-circular) objects is encountered in a number of practical instances e.g. flow of gases over heat exchanger tubes, wind flowing past transmission wires, flow past smoke stacks and chimneys etc. Uniform flow past a circular cylinder has great academic interest also because it is one of the simplest practical flows. D'Alembert's paradox for ideal flow, Stokes' solution for viscous creeping flows and Von Kármán's vortex street were all first stated with reference to this flow.

Theoretical Background

If a circular cylinder (in fact any closed object) is placed in a uniform stream (U) of an ideal fluid, then it is quite straightforward to show that the net force exerted by the fluid on the cylinder is zero, because the pressure distribution on the surface of the cylinder is symmetrical, both parallel to the stream

and perpendicular to it. This cannot be true for a real fluid D'Alembert's paradox.

For a real fluid the flow pattern and consequently the drag force depends greatly on the Reynolds number. If the magnitude of the uniform flow is, U , and the cylinder diameter is, D , then a convenient Reynolds number would be $Re = UD/\nu$. The drag force, F_D , is usually normalised by the dynamic pressure ($\frac{1}{2}\rho U^2$) yielding the drag coefficient, C_D , defined as $C_D = F_D/(\frac{1}{2}\rho U^2 DL)$, where, L , is the length of the cylinder. For very small Re , ($Re \ll 1$) the flow does not separate from the surface and the flow pattern is symmetric. However, the pressure distribution is not symmetric, giving rise to a pressure drag and there is, in addition, a large drag due to the viscous shear stress acting on the cylinder surface. At $Re \approx 4$ the pattern changes as the flow separates on the downstream side of the cylinder and the pressure drag increases significantly. As Re increases further the viscous drag continues to diminish in comparison with the pressure drag and for $Re \geq 1000$, the viscous drag is negligible.

Thus for large Re one can obtain the drag force experimentally by integrating the pressure distribution over the surface of the cylinder. Since the flow is symmetric about the top and bottom halves of the cylinder we have,

$$F_D = -2 \int_0^\pi p(\theta) \cos\theta L r d\theta \quad (9.1)$$

In order to measure the static pressure distribution the cylinder is equipped with a static pressure tap connected to an inclined manometer.

The drag force may also be determined by applying the momentum equation to the control volume shown in Fig 9.1. The flow at section 1 is uniform and may be determined with single velocity measurement. The flow at section 2 however, is not uniform and is determined with the help of a travelling pitot-static tube. We may assume that there is no mass flow through the top and bottom faces of the control volume (can be verified). Then if the average pressures at sections 1 and 2 are P_1 and P_2 respectively, we have,

$$F_D = \rho U^2 A - 2\rho L \int_0^b u^2(y) dy + (P_1 - P_2)A \quad (9.2)$$

where, A is the area of cross-section of the wind tunnel ($= 2bL$).

The velocity $u(y)$ is determined as (assuming $\rho_m \gg \rho$),

$$u(y) = \sqrt{2 \frac{\rho_m}{\rho} gh(y) \sin\phi} \quad (9.3)$$

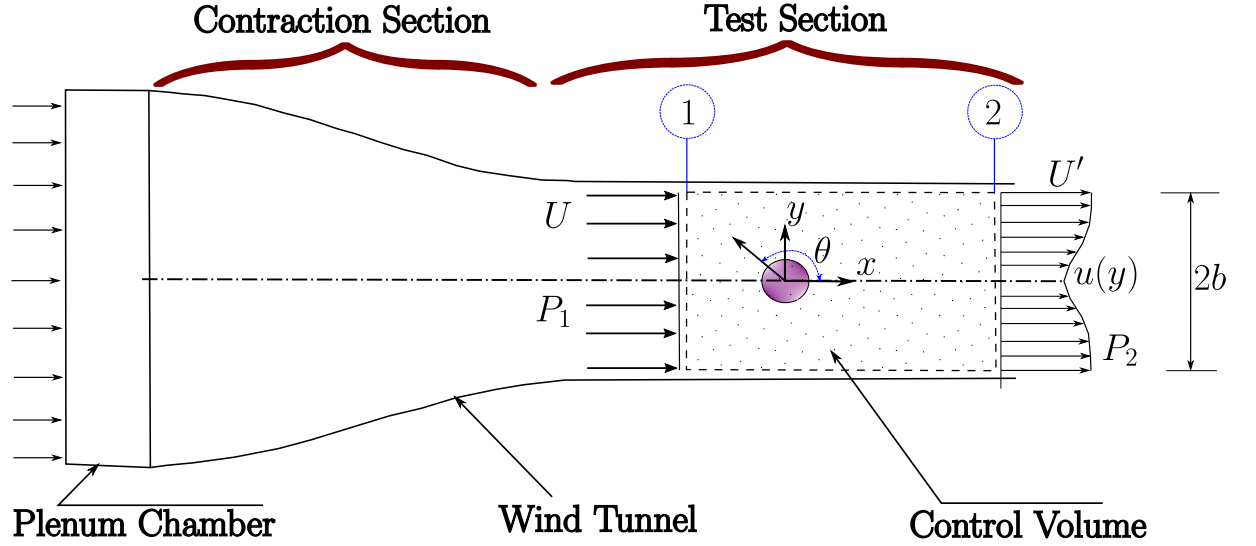


Figure 9.1: Schematic diagram of the Experimental set-up.

where, ϕ , is the angle of inclination of the manometer, h , is the deflection in the manometer and ρ_m is the density of the manometric fluid.

Experimental set-up

The rig consists of a wind tunnel consisting of a plenum chamber for smoothening out the flow, a contraction section for reducing turbulence levels and the boundary layer thickness, followed by the test section in which the circular cylinder is placed. The flow is driven by an axial flow fan. The outputs of the static pressure taps and the pitot-static tube are connected to a an inclined manometer bank. A schematic diagram is shown in Fig. 9.1.

Procedure

1. Measure the ambient temperature and atmospheric pressure. This will enable you to get the correct value for the density of air, ρ , used in the calculation of velocities.

2. Measure the angle of inclination of the manometers.
3. Measure the length, L , of the cylinder and the test section height, $2b$.
4. Start the axial flow fan. Take only one measurement of the velocity at section 1 since the flow is uniform there and move this pitot tube out of the way so its wake does not interfere with the flow past the cylinder.
5. Measure the velocity profile at section 2 using the travelling pitot-tube. Move the pitot-tube to the bottom wall so that it barely touches it. Move it away from the wall by approximately one diameter and take approximately 20 measurements between this point and the centre of the tunnel.

Note: Don't take the reading when the pitot-tube is touching the bottom wall.

6. Measure the static pressure distribution at the surface of the cylinder by rotating the cylinder to various angular positions.
7. Evaluate F_D by numerically integrating the pressure distribution and using the momentum equation and compare the two values of F_D and C_D .

Observations

1. Room Temperature = $^{\circ}C$.
2. Atm. Pressure = $mm\ Hg$.
3. Angle of inclination of manometer, $\theta =$ $^{\circ}$.
4. Length of the cylinder, $L =$ m .
5. Height of the tunnel, $2b =$ m .
6. Readings for the velocity at section 1.
 - (a) Total head $h_1 =$ m .
 - (b) Static head, $h_2 =$ m .
 - (c) Dynamic head, $h = h_1 - h_2 =$ m .
7. The Pressure Distribution at Table 9.1.
8. The velocity distribution (Table 9.2) at section 2.
9. Generate tables of $u(y)$ versus y and $p(\theta)\cos(\theta)$ versus θ .

Table 9.1: Pressure Distribution Table

Serial No.	Ang. Position	Static head	Reference head
	(degrees)	h' , (cm water)	h_0 , (cm water)
1			
2			
3			
4			
5			
6			
7			
8			
9			
10			
11			
12			
13			
14			
15			
16			
17			
18			
19			
20			

10. Plot $u(y)$ and $p(\theta)$.
11. Evaluate F_D and C_D by both methods.

Discussion

Highlight what you consider to be the key results of the experiment and discuss the results. List out the sources of errors and suggest methods for improving the experiment.

Table 9.2: Velocity Distribution Table

Serial No.	Vertical Position	Total head	Static head
	y (cm)	h_1 , (cm water)	h_2 , (cm water)
1			
2			
3			
4			
5			
6			
7			
8			
9			
10			
11			
12			
13			
14			
15			
16			
17			
18			
19			
20			

Experiment 10

Development of Turbulent Pipe Flow

Objective

To study the development of turbulent flow through a round duct.

Practical Relevance

When uniform flow enters a duct or when there is a sudden change in flow geometry due to, for example an obstruction, sudden expansion or contraction, elbow or tee joint, a flow control valve etc., it requires a certain length of straight pipe before the flow becomes fully developed. It is important to know this development length for flow measurement devices to be properly located in pipelines.

Theoretical Background

When uniform flow enters a constant area duct, the velocity at the walls immediately drops to zero due to viscous interaction between the fluid and the wall. As the flow proceeds, the region affected by the walls steadily increases, while the centreline velocity increases, to satisfy continuity. Finally shear is established over the whole flow, except the centre which is a point of symmetry, and the flow becomes fully developed i.e. the velocity profile

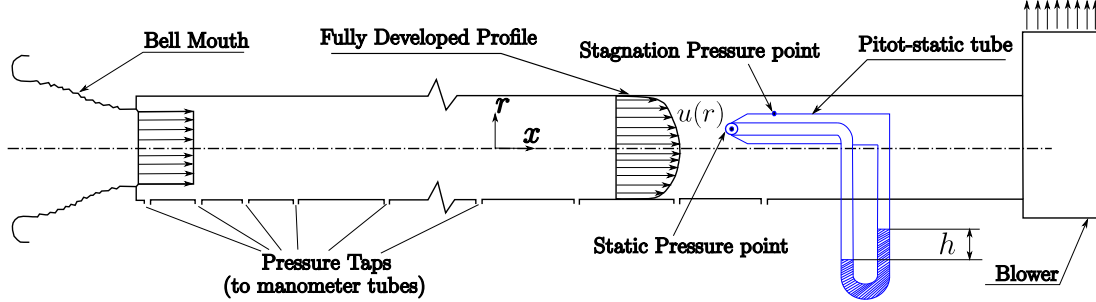


Figure 10.1: Schematic diagram of the Experimental set-up.

does not change with downstream distance. It is quite straightforward to show that when the flow is fully developed, the downstream mean pressure gradient is constant for both laminar and turbulent flows. However, the velocity profiles are very different for laminar and turbulent flows. In the laminar flow the velocity profile is parabolic, thus the velocity, u , at any radial position, r , may be expressed in terms of the centreline velocity, $u(0)$; and the pipe radius, R , as,

$$u(r) = u(0)\left(1 - \left(\frac{r}{R}\right)^2\right) \quad (10.1)$$

When the flow is turbulent, i.e. $Re (= 2UR/\nu$, where, U , is the avg. velocity) > 2000 , the profile is fuller (closer to uniform flow) because of enhanced transport of momentum to the wall due to turbulent motion and the mean velocity profile may be approximated by a power law as,

$$u(r) = u(0) \left(1 - \frac{r}{R}\right)^{\frac{1}{n}} \quad (10.2)$$

The value of n increases slowly with Reynolds number. For $Re = 10^5$, $n = 7$.

In the experiment the flow is turbulent and, as shown in Fig. 10.1, the flow undergoes development both due to the entry flow being approximately uniform and due the presence of a sudden expansion close to the entrance. By determining the wall static pressure development, the start of the fully developed region may be established. In addition, the flow rate may be obtained by

determining the velocity profile at any station after the flow is fully developed. Thus the experimentally obtained friction factor, $(\equiv (dp/dx)(4R/\rho U^2))$, may be compared with the Blasius correlation, $f = 0.316/((Re)^{1/4})$.

Experimental set-up

The rig consists of a long smooth pipe equipped with a bell-mouth ending in a sudden expansion, at one end. At other end of the pipe is connected to the suction side of a centrifugal blower. The pipe is fitted with pressure taps along its length and a pitot-static tube is used for velocity measurement. A schematic diagram is shown in Fig. 10.1.

Procedure

1. Measure the ambient temperature and atmospheric pressure. This will enable you to get the correct value for the density of air, ρ , used in the calculation of velocities.
2. Measure the angle of inclination of the manometers.
3. Note down the location of the pressure taps.
4. Start the blower and open the control valve at the exit completely.
5. Note down the manometer readings for all the pressure taps and plot pressure versus the downstream distance, x , to determine the start of the constant pressure gradient region.
6. Use the traverse gear to make the pitot-tube touch the opposite wall. Note down the scale reading and move the pitot-tube one diameter away from the wall. Take about 10 velocity measurements between this point and the centre of the duct to get the velocity profile.

Note: Do not take the manometer reading when the pitot-tube is touching the wall.

$$u(r) = \sqrt{2 \frac{\rho_m}{\rho} g h(r) \sin \phi}$$

where, ϕ , is the angle of inclination of the U-tube manometer with respect to the horizontal.

7. Plot $u(r)$ versus r , fair a smooth curve through the data points and extend the curve such that $u(R) = 0$. Numerically integrate $ru(r)$ by reading off the velocity at 20 equidistant radial locations and find the flow rate, Q , and the average velocity U . Evaluate the ratio $K = u(0)/U$.
8. Find the experimental friction factor as defined above and compare with that obtained from Blasius' formula.
9. Change the flow rate by closing the control valve completely. For this flow rate do the following:
 - (a) Take the readings of the pressure tap at the beginning of the constant gradient region as obtained above and the final pressure tap and evaluate the pressure gradient.
 - (b) Measure only the centreline velocity and obtain U and Q assuming that K is the same as for the previous flow rate.
 - (c) Evaluate f by the two different methods as above and compare them.

Observations

1. Room Temperature = _____ $^{\circ}C$.
2. Atm. Pressure = _____ $mm\ Hg$.
3. Angle of inclination of manometer set 1 (pressure drop), $\theta_1 =$ _____ $^{\circ}$.
4. Angle of inclination of manometer set 2 (velocity), $\theta_2 =$ _____ $^{\circ}$.
5. Diameter of the pitot tube, $d =$ _____ mm .
6. Diameter of the duct, $D =$ _____ m .
7. Flow rate 1: Pressure and velocity distributions (Table 10.1).
8. Plot $h(x)$ versus x , and determine the region of linear pressure gradient and the pressure gradient, dP/dx , there.
9. Plot $u(r)$ versus r , and determine the flow rate Q (using the procedure outlined in the handout), and the average velocity U . Also determine K , the ratio of the centreline velocity to the average velocity.
10. ~~Flow rate 2:~~
 - (a) ~~$\Delta h =$ _____ $cm\ water$.~~

Table 10.1: Observation Table

Sr. No.	Axial Position	Static head		Sr. No	Radial Position	Total Head	Static Head	Velocity
	$x, (m)$	$h(x), (cm \text{ water})$			$r (cm)$	$h_1, (cm \text{ water})$	$h_2, (cm \text{ water})$	$u(r), (m/s)$
1				1				
2				2				
3				3				
4				4				
5				5				
6				6				
7				7				
8				8				
9				9				
10				10				
11				11				
12				12				
13				13				
14				14				

(b) $\frac{dP}{dx} = \text{—————} N/m^3.$

(c) ~~Centerline velocity and average velocity:~~

i. $h_1 = \text{—————} cm.$

ii. $h_2 = \text{—————} cm.$

iii. $u(0) = \text{—————} m/s.$

iv. $U = u(0)/K = \text{—————} m/s.$

Discussions

Highlight what you consider to be the key results of the experiment and discuss the results. List out the sources of errors and suggest methods for improving the experiment.

Bibliography