Southern Illinois University Carbondale

ENGR 370

ENGINEERING FLUID MECHANICS

LABORATORY MANUAL -

updated: Dr. John Nicklow, P.E., P.H.
College of Engineering
(November 3, 2010)

Table of Contents

LIST OF FIGURES

INTRODUCTION

This laboratory manual is intended to guide you through several experiments in the study of engineering fluid mechanics. Because of the nature of the course and laboratory facilities, you may be required to perform some experiments on material not yet covered in the classroom. This requires an extra effort on your part to read the relevant sections of the textbook, as well as the lab manual, *before* you come to the lab. Being prepared will assist you in understanding the experimental work and allow you to finish in the allotted time.

The lab report is considered to be an engineering technical report. As such, you will be evaluated on your ability to correctly complete the experiment and analysis, as well as your ability to clearly communicate your methodology, results, and ideas to others. The text should be completed using a word processor. Write in the past tense, and pay special attention to spelling and grammar. Be sure to cite references for any information, including equations, which you did not originate. Plagiarism, which involves the copying of published or unpublished work of another individual, will be severely penalized (please refer to “Academic Integrity” in the course syllabus). All charts, plots and drawings should be original (e.g. created by you) and completed using a computer. Most charts, plots and drawings can be imported directly from the software used to create them into your word processing software. The requirements for each section of the report are explained in detail on page 4, along with the criteria upon which your grade will be based. Your teaching assistant will distribute an example laboratory report, which should be useful as a guide.

Your ability to work as a team will be very important in the laboratory. Keep in mind that you do have a time constraint and need to be both accurate and efficient in your work. Try identifying the responsibilities of each group member before you begin an experiment. Some responsibilities include recording the data, obtaining measurements, adjusting an apparatus between measurements, and distributing copies of the data to each group member after the experiment has been completed.

REPORT REQUIREMENTS

Each student must submit a lab report consisting of the following items:

a.	Title Page	Include the course number, section number, and course title, descriptive title of the experiment, date completed, date submitted, and *prepared by* (author's name) and *submitted to* (lab instructor's name)	(5)
b.	Table of Contents	List major report headings;	(1)
		Show a *list of tables and figures* with titles;	(1)
		Indicate the contents of the appendix ;	(1)
		Show corresponding page numbers	(1)
c.	Objective	Clearly state the purpose of the investigation	(5)
d.	Theory	Discuss fundamental principles behind the experiment;	(6)
		Clarify equations to be used and define all variables;	(3)
		Indicate important assumptions for validity of equations;	(2)
		Indicate your anticipated results;	(5)
		Cite references for borrowed ideas or expressions	(2)
e.	Apparatus	Drawings of the apparatus with *important* dimensions	(5)
f.	Procedure	A concise list of the steps *your group* performed, not necessarily what you were instructed to do	(5)
g.	Results	All your *results* should be presented here; Tables, charts and plots should be incorporated into the report text; For each table or figure, provide a brief narrative that explains the significance of the table or figure, its content, and, by referencing equations in the "Theory" section, how values were obtained; Provide a number and title for each table and figure that corresponds to that given in (b)	(25)
h.	Discussion	Discuss the accuracy and precision of *each* result;	(15)
		Analyze probable sources of error	(5)
i.	Conclusion	State and justify whether your initial objective was met; Can you make recommendations for future related work	(8)
j.	Appendix	List of references cited in the report; Original, handwritten data tables; Sample hand calculations (optional, but beneficial)	(5)
		TOTAL POINTS	(100)

EXPERIMENT 1
PROPERTIES OF FLUIDS

- *Stoke's Law*
- *Viscosity & Drag*

I. Introduction

The transport and accumulation of sediment in waterways and reservoirs, the movement of dust and other pollutant particles in the atmosphere, and the flow of liquids through porous media are examples of phenomena in which drag forces play important roles. Drag forces can be expressed as functions of a particle's size and velocity, as well as properties of the surrounding fluid

Consider a sphere that is dropped into a column of fluid. The sphere will eventually reach a constant **terminal velocity** at which acceleration has ceased. The surface and body forces acting on the sphere are the drag force, F_D, buoyant force, F_B, and gravity force, W.

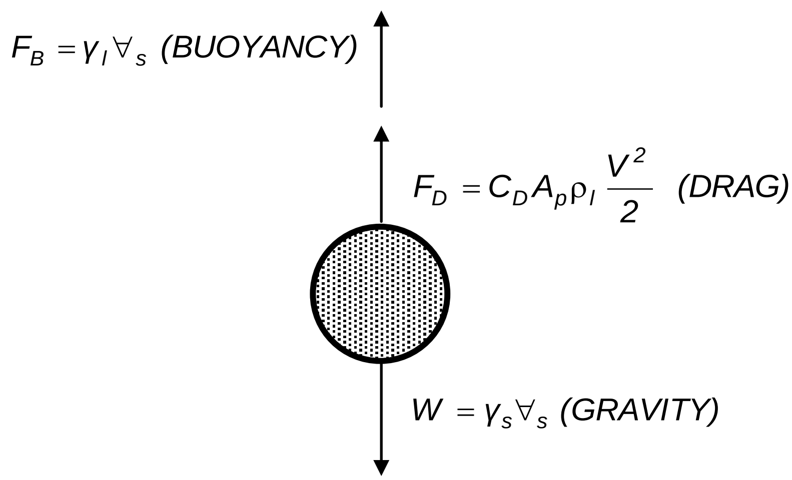

Figure 1: Free Body Diagram of Falling Sphere

Where V is the sphere fall velocity, γ_s and γ_l are specific weights of the sphere and surrounding fluid, respectively, C_D is the **coefficient of drag**, and ρ_l is the mass density of the fluid. Using d as the diameter of the sphere, and given that the volume of the sphere, $\forall_s$, is $\pi d^3/6$ and the projected area of the sphere, A_p, is $\pi d^2/4$, a summation of vertical forces yields

(i) $$\sum F = 0 = \left(C_D \frac{\pi d^2}{4} \frac{\rho_l V^2}{2} \right) + \left(\gamma_l \frac{\pi d^3}{6} \right) - \left(\gamma_s \frac{\pi d^3}{6} \right)$$

An expression for the drag coefficient can be obtained by solving equation (i) for C_D as follows

(ii) $$C_D = \frac{4d(\gamma_s - \gamma_l)}{3\rho_l V^2}$$

Thus for a sphere moving through a known fluid, the sphere's velocity may be measured and used to compute the coefficient of drag. Similarly, solution to (i) can yield the dynamic, or absolute, viscosity, μ.

(iii) $$\mu = \frac{d^2(\gamma_s - \gamma_l)}{18V}$$

The sphere's velocity can also be used to compute the **Reynolds number**, given as

(iv) $$Re = \frac{\rho_l V d}{\mu} = \frac{Vd}{\upsilon}$$

where υ is the kinematic viscosity of the fluid, respectively. For viscous fluids and a low velocity (i.e. low Reynolds number), the drag force has been shown to be

(v) $$F_D = 3\pi\mu V d$$

which is known as **Stoke's Law**. Comparing the general drag force expression, given in Figure 1.1, with Stoke's Law, it follows that for highly *laminar flows*, or flows having *Re* <1,

(vi) $$C_D = \frac{24}{Re}$$

In logarithmic form, equation (vi) is

(vii) $$log\,(C_D) = -log(Re) + log(24)$$

Note that this expression has the form of a linear equation, $y=mx+b$. Ideally, a logarithmic plot of the C_D vs. *Re* will then yield a straight line having a slope of (-1) measured with a *linear* scale.

II. Objective

The objective is to demonstrate the mechanics of fluid drag on a spherical object during free fall for at least two different viscosities and to evaluate the relationship between the coefficient of drag and the Reynolds number.

III. Procedure

1. Obtain spheres from the instructor.
2. Measure the diameter and weight of each sphere in order to evaluate each sphere's specific weight and specific gravity.
3. Obtain values for the specific gravity of the glycerin and 30-wt. motor oil from the lab instructor.
4. Carefully place one sphere in one of the tubes. The sphere should be completely immersed before it is released.
5. Obtain the fall velocity by using a digital timer and recording the time required for the sphere to fall a known distance.
6. The sphere should be removed via the double valve at the bottom of the tube.
7. Repeat steps (2) – (5) for at least six spheres that vary in size and weight.
8. Repeat steps (2) – (6) for the second tube.

IV. Results

1. For each trial, compute the drag coefficient, C_D, dynamic viscosity of the fluid, μ, and the Reynolds number, *Re*, for the laboratory data.
2. Construct a *logarithmic* plot of *Re* (abscissa) of the fluid vs. C_D (ordinate) of the spheres.
3. Determine the equation of the line created in step (2) and the associated R^2 value, and show them on the graph. Discuss the extent to which the equation matches equation (vi) derived from Stokes Law? [Note: Recall when evaluating the equation that it is a logarithmic plot. Therefore, you may wish to create a temporary, arithmetic plot of (log *Re*) vs (log C_D), and determine the equation of that plot using linear regression. In Microsoft Excel©, a linear trendline can be created, then the R^2 value, which represents the "goodness" of fit, and equation of the line can be automatically inserted on the plot. For additional information regarding regression analysis and the R^2 statistical parameter, students can refer to http://civil.engr.siu.edu/nsflab/Manuals/Main_General.htm].
4. *Quantitatively* compare the mean dynamic viscosity for both the glycerin and oil with corresponding values obtained from an authoritative source, such as your textbook. [Note: Percentage of error can be expressed by computing {(Theoretical – Experimental Value)/(Theoretical Val)} x 100%]
5. What conclusions can you draw from the results obtained for the two different liquids?

EXPERIMENT 2
BERNOULLI'S EQUATION

- *Validation of Bernoulli's Concept*
- *Static & Stagnation Pressure*

I. Introduction

The Bernoulli equation is one of the most widely used, and misused, equations in the analysis of fluid flow. It attempts to relate the pressure, elevation, and velocity between any two points in an **inviscid** (ideal), **constant density** fluid flowing at **steady state**. Consider a differential fluid element aligned *along a streamline*. A streamline is a line drawn in the flow field in such a manner that the velocity vector at each and every point on the streamline is tangent to the streamline at any instant. For this case, s is the direction of the streamline, Δz is the vertical height of the fluid element, ΔA is the cross-sectional area of the element, p is the pressure, V is velocity of the fluid, W is the weight of the fluid element, and θ is the angle that the element makes with the horizontal.

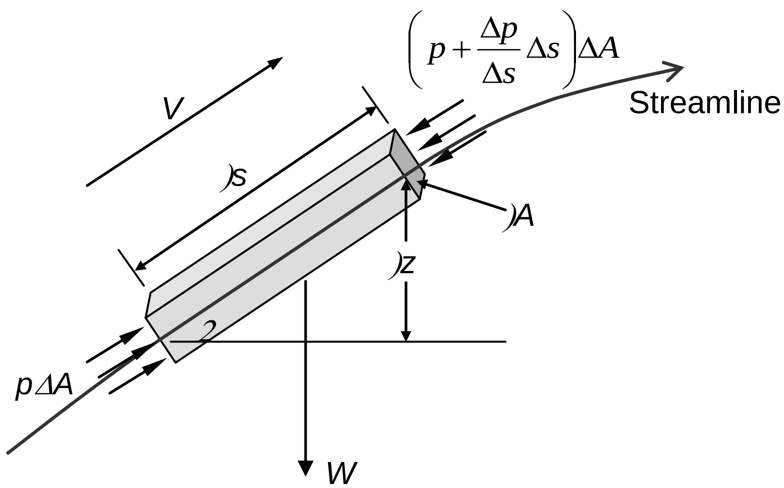

Figure 2: Differential Fluid Element and Streamline

Application of Newton's second law in the direction of flow yields

(i) $$\sum F_s = ma_s \quad \text{OR} \quad p\Delta A - \left(p + \frac{\Delta p}{\Delta s}\Delta s\right)\Delta A - \gamma \Delta s \Delta A \sin\theta = \rho \Delta s \Delta A a_s$$

where F_s represents the external forces acting along the streamline, m is the mass of the fluid element, a_s is the acceleration of the fluid element along the streamline, and γ and ρ are the specific weight and density of the fluid, respectively. Noting that $\sin\theta$ is equivalent to $\Delta z/\Delta s$, a little algebraic manipulation yields

(ii) $$-\frac{\Delta p}{\Delta s} - \gamma\frac{\Delta z}{\Delta s} = \rho a_s$$

In the limit, as Δs approaches zero,

(iii) $$-\frac{\partial p}{\partial s} - \gamma \frac{\partial z}{\partial s} = \rho a_s$$

which is known as Euler's equation of motion. Also consider that since velocity is a function of time and position along the streamline, acceleration along a streamline can be expressed as

(iv) $$a_s = \frac{\partial V}{\partial t} + V \frac{\partial V}{\partial s}$$

For steady flow, the time dependent term approaches zero, and (iv) reduces to

(v) $$a_s = V \frac{\partial V}{\partial s}$$

Substituting (v) into (iii), along with application of the product rule, yields

(vi) $$-\frac{\partial p}{\partial s} - \gamma \frac{\partial z}{\partial s} = \frac{\rho}{2} \frac{\partial V^2}{\partial s}$$

If density is held constant, integration over *s* gives

(vii) $$p + \gamma z + \frac{\rho V^2}{2} = \textit{Constant along a streamline}$$

Finally, dividing by the specific weight yields the **Bernoulli equation**

(viii) $$\frac{p}{\gamma} + z + \frac{V^2}{2g} = \textit{Constant} = H$$

where *H* is total energy head. The first term on the left hand side of the Bernoulli equation is known as the pressure head, the second is the elevation head, and the third is the velocity head. Each of these terms has units of energy (ie. N-m/m, ft-lb/lb), which yields a dimension of length. The equation, therefore, provides a relation between pressure, elevation and velocity for any two points in a flow field that is frictionless, steady, incompressible and homogeneous. It is also frequently expressed as

(ix) $$\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g} = \frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g}$$

If the flow system is horizontal, the elevation differences can be neglected and the Bernoulli equation then reduces to

(x) $$\frac{p_1}{\gamma} + \frac{V_1^2}{2g} = \frac{p_2}{\gamma} + \frac{V_2^2}{2g}$$

II. Objective

The objective is to evaluate the validity of the Bernoulli equation by determining if the summation of pressure head and velocity head is constant at several locations in a horizontal duct.

III. Procedure

1. Measure the interior diameter and compute the cross-sectional area of the test section at each of the pressure tap locations.
2. Measure the distance from the upstream end of the test section to each of the pressure tap locations.
3. With the bench control valve and outlet control valve closed, start the pump. If needed, bleed any air from the interior of the manometers.
4. Very gradually open both valves and adjust them such that the fluid level in each of the manometers can be clearly read using the permanent scale.
5. Determine the flowrate by closing the ball valve in the volumetric tank and measuring the time required to accumulate a known volume of fluid in the tank. Use a digital timer and the sight glass located on the side of the hydraulic bench.
6. With the stagnation probe retracted, but not completely withdrawn, from the test section, record the fluid level in each of the first five test section manometers. Each manometer measures static pressure head at its tapping point, and the corresponding scale is in mm of water.
7. Traverse the stagnation probe along the length of the test section, and at each tap, record the level in the associated manometer, or the stagnation pressure head. The stagnation head represents total energy head since the fluid velocity at the stagnation point is zero.
8. Repeat steps (4) - (6) for at least one additional flowrate.

IV. Results

1. For each trial, use the values of flowrate and cross-sectional area to compute flow velocity and velocity head at each pressure tap location. To assist you in computing velocities, a sketch of the Venturi with dimensions is provided on the following page.
2. For each flowrate, determine the total energy head at each location by summing the static pressure head and velocity head.
3. On one drawing, plot the total energy head computed in (2), the stagnation pressure head, the static pressure head, and velocity head vs. approximate distance from the upstream end of the test section.
4. Repeat step (3) for the second flowrate.
5. *Qualitatively* compare the values for total head obtained from (2) with those obtained by measuring stagnation pressure.
6. Discuss the validity of the Bernoulli equation for the flow of water through the horizontal test section. As justification, make reference to the applicability of the assumptions that are made in the derivation of the equation.

Tapping Position	Manometer Legend	Diameter (mm)
A	h_1	25.0
B	h_2	13.9
C	h_3	11.8
D	h_4	10.7
E	h_5	10.0
F	h_6	25.0

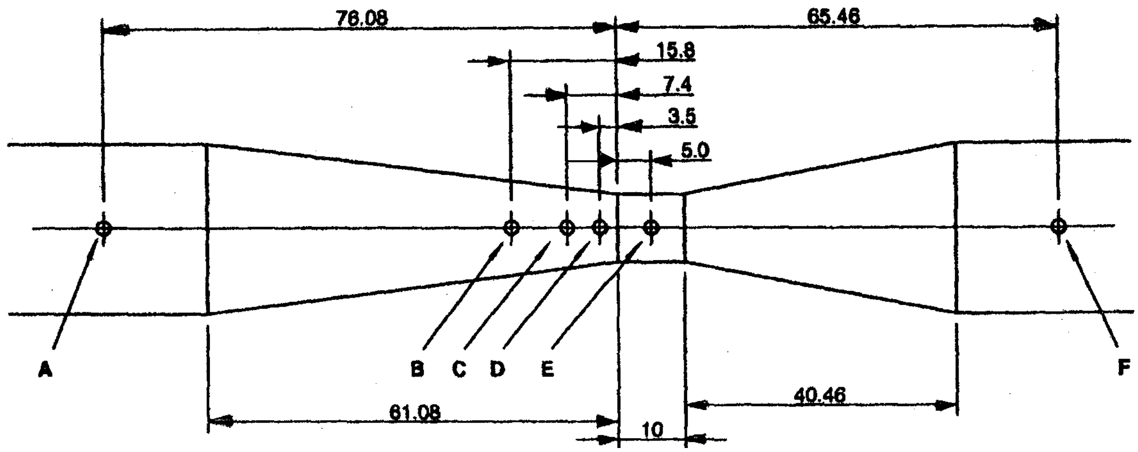

Figure 3: Venturi Dimensions (mm)

EXPERIMENT 3
SHARP-CRESTED WEIR

- *Flowrate Measurement*
- *Weir Equations*

I. Introduction

Determination of the flowrate of water in open channels is of significance in many aspects of society. For example, urban and industrial water supplies must be measured so that demands are satisfied; the amount of water required for the dilution of pollutants being wasted into a river can be calculated mathematically, but metering devices are required to measure the supplied flow; and the probable damages that result from floods can be determined by correlating the depth of water passing over a dam spillway, which is a special type of weir, to the volume of water flowing downstream.

Weirs for measuring purposes are usually of more simple and reproducible form than a dam spillway. *Sharp-crested weirs* are constructed using a smooth, vertical, flat plate with the upper edges sharpened. A popular weir for use in low flow situations is the V-notch weir, which has a sharp, V-shaped crest that extends over a partial width of the channel in which it is placed. When these weirs are designed so that at least a part of the flow passes over the V-notch, the equation for determining volumetric flowrate, neglecting the velocity head of the upstream flow, is:

(i) $$Q = CH^n$$

where C is a **coefficient of discharge** and is a function of the notch angle, θ, and n is a weir constant. The term H is the depth of water, or head, above the weir crest and is measured upstream of the crest a distance equal to approximately four times the head. The constants C and n must be determined experimentally for a given weir. In logarithmic form, the weir equation is

(ii) $$\log Q = n \log H + \log C \qquad OR \qquad \log H = \frac{1}{n}\log Q - \frac{\log C}{n}$$

Note that this expression resembles a linear equation having the form y=mx+b. To find the values of C and n, first create a weir calibration curve by plotting Q (abscissa) vs. H (ordinate) on logarithmic scales. Note that the slope on the logarithmic plot, measured with a *linear* scale, is then (*1/n).* One could also find n directly by determining the equation of the calibration curve or by computing the slope of the line from

(iii)
$$n = \frac{\log \frac{Q_1}{Q_2}}{\log \frac{H_1}{H_2}}$$

where the subscripts *1* and *2* refer to two representative points along the calibration curve. Once n is known, C can be determined using the equation of the curve or using measured data and equation (i). For V-notch weirs constructed to standard specifications, the value of both C and n is 2.5.

(iv)
$$Q = 2.5H^{2.5}$$

These values for C and n are only valid for cases when the assumptions of the equation's derivation are confirmed. These assumptions are that the weir face is smooth, the crest is sharp so that the flow does not drag in passing over the weir, atmospheric pressure exists below the lower nappe of the flow, velocity head is negligible, and viscous and surface tension forces are small.

II. Objective

The objective is to demonstrate the operation of a V-notch weir as a flow-measuring device and develop a weir calibration curve for a 90° V-notched weir.

III. Procedure

1. Start the pumps and establish a steady flow at a maximum level, or about 2 inches below the sidewalls to avoid overtopping.
2. Measure the head on the weir
3. Using a bucket of known weight and a timer, collect all water spilling over the weir in a reasonable amount of time. Volumetric flowrate can be evaluated by weighing the collected volume and subsequently dividing the weight by the specific weight of water and the amount of time that was required to collect the water.
4. Repeat steps (2) and (3) for at least six different head levels and flowrates by closing the discharge valves so as to produce reasonable intervals of head.

IV. Results

1. On the same graph, construct logarithmic plots of discharge, Q (abscissa), vs. head on the weir, H (ordinate), from (a) the laboratory data, and (b) weir formula calculations – use the laboratory data for H and solve for Q using equation (iv).
2. For the laboratory data, determine the equation of the line created in step (1a) and its associated R^2 value, and show them on the graph. What are the C and n values for the weir? [Note: Recall when evaluating the equation that it is a

logarithmic plot. Therefore, you may wish to create a temporary, arithmetic plot of (log *Q*) vs. (log *H*), and determine the equation of that plot using linear regression. Remember that the slope of this plot will be *1/n*. In Microsoft Excel©, a linear trendline can be created, then the R^2 value, which represents the "goodness" of fit, and equation of the line can be automatically inserted on the plot. For additional information regarding regression and statistical analysis, see http://civil.engr.siu.edu/nsflab/Manuals/Main_General.htm]

3. *Quantitatively* compare the coefficient of discharge and exponent obtained in step (2) with those used in step (1b) for weirs constructed of standard specifications, and discuss reasons for any discrepancies.

EXPERIMENT 4
PELTON WHEEL

- *Hydraulic Machines*
- *Efficiency and Power*

I. Introduction

A hydraulic machine is a device that receives energy at one point and converts that energy to useful work at another point. Examples of hydraulic machinery include impulse turbines, which extract useful work from fluid energy, and pumps, blowers and turbocompressors, which add energy to fluids. A specific type of impulse turbine that is used in many practical applications is a **Pelton wheel**.

Since there are always losses in machinery, the work output is always less than the energy input. The **efficiency**, e_f, of a turbine is defined as the ratio of the work output to the energy input from a fluid in terms of the time rate of energy exchange, or **power**.

(i) $$e_f = \frac{P_o}{P_i}$$

where P_i is the power input and P_o is the power output. The power supplied by a fluid is given as

(ii) $$P_i = \gamma Q \Delta H$$

where the term γ is the specific weight, Q is the volumetric flow rate, and ΔH is the total differential head. Output power can be evaluated by observing system load. Consider a resultant force, F, that is an indicator of output load and is tangent to an output pulley at a radius, r. As the shaft turns through an angle, θ, the work done, W_o, against the force is ($F \times s$), where s is the arc length. In radians, $s = r\theta$, and

(iii) $$W_o = Fr\theta$$

Since torque, T, is ($F \times r$) and θ is 2π radians, then

(iv) $$W_o = 2\pi Fr = 2\pi T$$

The time rate of work output is the power output. Therefore, $P_o = \omega T$, where ω is the angular velocity. In terms of the shaft speed, N, in revolutions per minute,

(v) $$P_o = \frac{\pi NT}{30}$$

In the case of a prony brake, the torque is found by measuring the applied force at a radius of 0.526 ft., so that equation (v) reduces to

(vi) $$P_o = \frac{\pi NF(0.526)}{30}$$

The efficiency of a turbine can then be determined using equations (i), (ii) and (vi).

II. Objective

The objective is to observe the operation of an impulse turbine and evaluate the variation of its efficiency as a function of output speed.

III. Procedure

1. Set up the equipment (Gilkes Tutor-Pelton Assembly with prony brake, gages, pump, pelton wheel, and tachometer):
   (a) Prior to starting the pump, the upper valve should be opened from two and one half to five turns. Record the number of turns of the valve.
   (b) Turn the electrical switch to "on" with the electric motor set to zero. Turn the motor control to a setting between 80 and 100 so that a pressure head between 40 ft. and 70 ft. is obtained.
   (c) Check that for the spear valve setting, the head and the flow rate remain constant between no load and full load conditions.

2. To obtain shaft speed (N) and efficiency (e_f) data, the input power should be set to a fixed value. Record the shaft speed, output load, constant differential head and flowrate for the experiment. Volumetric flowrate can be found by weighing a volume of collected water, dividing the weight by the specific weight of the water, and dividing again by the amount of time that was required to collect the water.
3. Vary the output load, F, and repeat step (2) for approximately 10 trials.

IV. Results

1. Determine the efficiency of the turbine for each trial.
2. Plot the shaft speed in rpm (abscissa) vs. the turbine efficiency (ordinate), and estimate the shaft speed at maximum efficiency.
3. Compute the power output at maximum efficiency in units of horsepower.

EXPERIMENT 5
IMPACT JET

- *Momentum Principle*
- *Surface Forces*

I. Introduction

Consider a jet of water striking a stationary plate as shown below. The jet is deflected with a resulting exchange in momentum. From Newton's second law of motion, the **momentum flux** in the control volume equals the magnitude of the net reaction exerted by the plate.

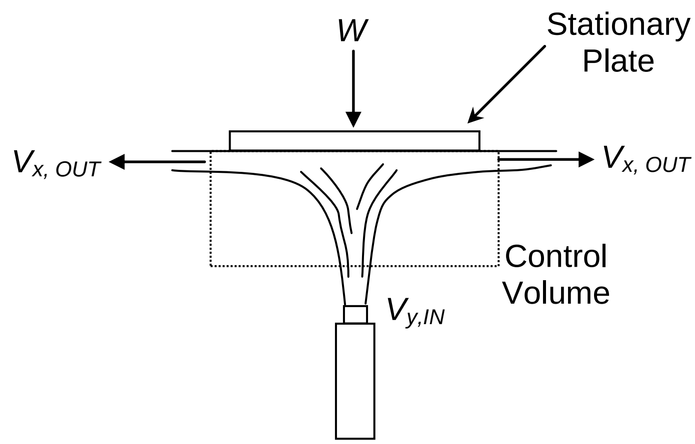

Figure 4: Liquid Jet Deflected by a Stationary Plate

Here it is assumed that the pressure in the streams that are leaving the control volume is equal to that entering the control volume. It is also assumed that surface resistance of the plate does not appreciably affect the velocity of the jet. If the control volume is drawn so that only the jet is included, the linear momentum equation can be applied to determine the reactive force on the plate. A summation of surface forces in the vertical direction yields

(i) $$\sum F_y = (\rho Q V_y)_{OUT} - (\rho Q V_y)_{IN}$$

where F represents surface and body forces, ρ is the mass density, Q is the volumetric flowrate and V_y denotes the velocity in the vertical direction. If a force W is applied to the plate and transmitted to the jet as a resistance, then

(ii)

$$-W = 0 - \left(\rho Q V_y\right)_{IN}$$

or

(iii)

$$W = \left(\rho Q V_y\right)_{IN}$$

II. Objective

The objective is to calculate the reactive force on a plate by means of the linear momentum equation, and to compare computed results with observed values.

III. Procedure

1. Open the discharge valve and turn on the electrical switch to start the pump motor.
2. Fill the tank with water and record the diameter of the nozzle as 0.40 inches.
3. Once a steady state condition has been reached, record the time required to fill the 22” x 11.75” section of the tank to a particular depth. Using the tank dimensions, depth of water in the tank, and elapsed time, the volumetric flowrate can be computed.
4. Pour a small amount of lead shot, to be used as the applied force (*W*), into the designated cup and place the cup on the spring apparatus. The corresponding experimental reactive force is found by weighing the cup and the lead shot.
5. Use the pump valve to incrementally increase or decrease the flow rate, and repeat steps (3) and (4) for approximately ten trials.

IV. Results

1. Compute the discharge velocity from the nozzle various applied weights.
2. Calculate the theoretical reactive force, *W*, using the linear momentum equation.
3. *Quantitatively* compare the experimental and calculated values of the reactive force by computing percentage of error incurred.

EXPERIMENT 6
CLOSED CONDUIT FLOW

- *Reynolds Number*
- *Friction Factor*
- *Energy Losses*

I. Introduction

The losses of energy in conduits flowing full of a liquid usually result from the resistance of the conduit walls to the flow, or from pipe appurtenances (e.g. elbows, contractions, valves) that cause the flow velocity to be changed in magnitude and/or direction. These energy losses must be calculated so that, for example, the proper size and number of pumps can be specified in the design of a municipal water distribution system; the conduit size for a gravity-flow urban drainage project may be determined; or the optimal size of valves and the radius of curvature of pipe elbows can be stipulated in the specifications of a pipeline design.

When the ratio of the length of a pipeline, *L*, to its diameter, *D*, exceeds 2000:1, or when pipe appurtenances are not present in a pipe, energy losses, also called **head losses**, are predominantly the result of internal pipe friction. The remaining losses that result from pipe appurtenances are termed *minor* losses.

To determine the head loss for a pipe system, consider the energy equation written between two sections of the pipe,

(i)
$$\frac{p_1}{\gamma}+\frac{V_1^2}{2g}+z_1=\frac{p_2}{\gamma}+\frac{V_2^2}{2g}+z_2+h_L$$

where *z* is the elevation of the centerline of the pipe relative to an arbitrary datum, *V* is flow velocity, *g* is the gravitational constant, *p* is pressure at the centerline of the pipe, γ is the specific gravity of the fluid, and h_L is the total energy loss between sections 1 and 2, which is limited to frictional losses only, h_f, when minor losses are negligible. The velocity in equation (ii) can be evaluated if the flowrate and pipe dimensions are known. If the pressure at sections 1 and 2 can be measured, the energy equation can then be used to evaluate the unknown head loss through the pipe.

A second method for evaluating head loss employs the **Darcy-Weisbach** equation and the **Moody diagram**. The former can be used to directly evaluate energy loss caused by pipe friction and is expressed as

(ii)
$$h_f=f\,\frac{L}{D}\,\frac{V^2}{2g}$$

where f is the dimensionless Darcy-Weisbach friction factor, L is the length of pipe section, and D is the pipe diameter.

The Moody diagram (see course text) illustrates the relationship between the friction factor, f, the Reynolds number, Re, and relative roughness of the pipe. On this diagram there are three zones of flow; laminar, transitional, and turbulent. Following the Moody diagram, the following equations can be used to determine a friction factor, f, for *smooth* pipes, where f is a function of the Reynolds number only, given as

(iii)
$$Re = \frac{VD}{\upsilon}$$

where υ is the kinematic viscosity of the fluid.

(iv) Laminar Flow (Re < 2000) $$f = \frac{64}{Re}$$

(v) Transitional Flow (2000 < Re < 3000) $$f = \frac{0.316}{Re^{0.25}}$$

(vi) Turbulent Flow (Re > 3000) $$\frac{1}{\sqrt{f}} = 2.0\log(Re\sqrt{f}) - 0.8$$

The energy loss caused by pipe friction may also be expressed as

(vii)
$$h_f = ZLV^n$$

where the constants Z and n are function of the type of flow (i.e., laminar, turbulent, or transitional). This equation in logarithmic form is

(viii)
$$\log h_f = n\log V + \log ZL$$

Note that this expression has the form of a linear equation, y=mx+b. To determine the constants Z and n, first plot V (abscissa) vs. h_f (ordinate) on logarithmic scales. The slope on the logarithmic plot, measured with a *linear* scale, is then n. An alternative method of finding n involves determining the equation of the curve or by computing the slope of the line directly using the formula

(ix) $$n = \frac{\log \frac{h_{f1}}{h_{f2}}}{\log \frac{V_1}{V_2}}$$

where the subscripts *1* and *2* refer to representative points along the curve. Once *n* is known, *Z* can be evaluated using the equation of the curve or using measured data and equation (vii).

The **energy grade line** (EGL) is a visual representation of Bernoulli's equation applied to a sequence of locations in the pipe. The distance from an arbitrary datum to the EGL at any single point is the total energy head, *H*, represented as

(x) $$\frac{p}{\gamma} + \frac{V^2}{2g} + z = H \qquad \textit{(Bernoulli Equation)}$$

The **hydraulic grade line** (HGL) represents the piezometric head, or the sum of the pressure and elevation terms of the Bernoulli equation. Thus, the difference between the HGL and EGL represents the velocity head, $V_2/2g$, and the slope of the EGL is the ratio of head loss to pipe length, as indicated below for a gradual expansion on a downward slope.

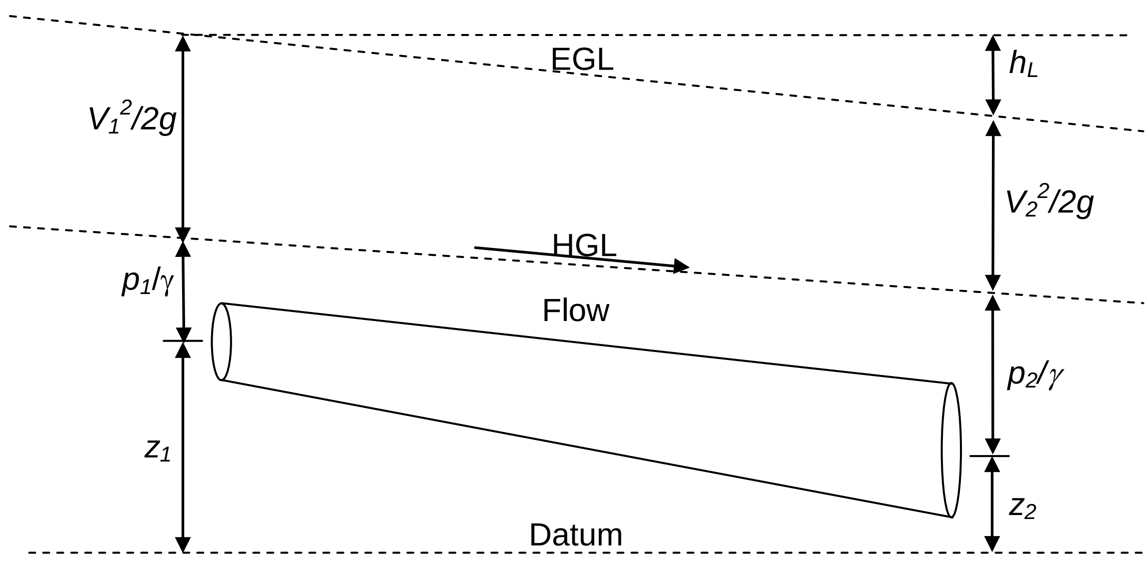

Figure 5: Energy and Hydraulic Grade Lines

II. Objective

The objective is to study steady flow in a closed conduit and evaluate energy losses through a pipe system.

III. Procedure

A general description of the closed system is as follows:

1. Flow is directed from the reservoir tank through a 1.025 in. I.D. pipe.
2. A differential manometer gage is connected to two pressure taps located on the pipe to allow a measurement of head loss. The pipe section between the taps is 4.0 ft. in length.
3. The flow travels through a flow-metering device, which provides a digital display of volumetric flowrate.
4. By the definition of a closed system, flow is recycled back to the reservoir.

To perform the experiment:

1. Turn on the pump and check that the flow is following the correct path. Make any necessary adjustments to the differential manometer gage.
2. Set the flowrate at a maximum rate, and once the flow has stabilized, record the head loss across the pipe length and the corresponding flowrate.
3. Reduce the flowrate in incremental steps, being sure to record the head loss at each flowrate, until the manometer readings are too small to accurately determine.

IV. Results

1. Calculate the Reynolds number using equation (iii) and then the friction factor from equation (iv), (v) or (vi). What type of flow is occurring within the pipe?
2. Using the results from (1) and the Darcy-Weisbach equation, compute the theoretical head loss for each flowrate. *Quantitatively* compare the theoretical values to your measured head loss data obtained using the differential manometer. Discuss your results.
3. Determine the values of *Z* and *n* for the conditions of this experiment. First, use equations (ii), (iii) and (vii), along with either equation (iv), (v), or (vi), to solve for *Z* and *n* algebraically. Next evaluate both constants by using your laboratory data and one of the methods described in Section 1. *Quantitatively* compare the theoretical and experimental values of *Z* and *n*. [Note: If determining the equation of the curve for laboratory data, recall that this is a logarithmic plot. Therefore, you may wish to create a temporary, arithmetic plot of (log *V*) vs. (log h_f), and determine the equation of that plot using linear regression. In Microsoft Excel©, a linear trendline can be created, and the R^2 value, which represents the “goodness” of fit, and equation of the line can be automatically inserted on the plot. For additional information regarding regression and the R^2 statistical parameter, refer to http://civil.engr.siu.edu/nsflab/Manuals/Main_General.htm]
4. Draw the energy and hydraulic grade lines for the maximum flowrate. *Qualitatively* indicate elevation and pressure heads, but numerically identify the velocity head component of total energy and the head loss over the length of pipe.

EXPERIMENT 7
HYDRAULIC JUMP

- *Jump Phenomenon*
- *Specific Energy and Force*

I. Introduction

A schematic of a hydraulic jump in an open channel of small slope is shown below. In engineering applications, the hydraulic jump frequently appears downstream from spillways or sluice gates where flow velocities are high. It may be used as an effective dissipater of kinetic energy, and thus prevent scour of an alluvial river bottom, or can be induced as a mixing device in water or sewage treatment designs. In design applications, the engineer is concerned primarily with predicting the occurrence, size, and location of the jump

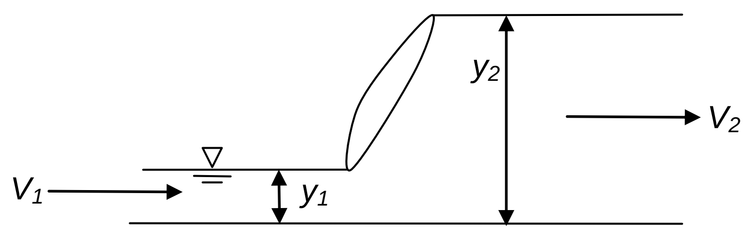

Figure 6: Hydraulic Jump

A hydraulic jump is formed when liquid at high velocity discharges into a zone of lower velocity, creating a rather abrupt rise in depth. The rapidly varying water surface is typically accompanied by violent turbulence, eddying, air entrainment, and surface undulations. The high velocity flow is known as **supercritical**, and occurs at depths below critical depth, whereas the low velocity flow is **subcritical**, and occurs at depths greater than critical depth. The **critical** depth, y_c, is the depth associated with the point of minimum energy in the associated control volume and is an unstable depth that occurs within the jump. A parameter that further can be used to characterize critical flow is the **Froude number**, *Fr*, expressed as

(i)
$$Fr = \frac{V}{\sqrt{gD}}$$

where V is the flow velocity, g is the gravitational acceleration, and D is the hydraulic depth, defined as a ratio of flow area to top width at a given location. For a channel of rectangular cross section, a discharge, Q, and constant width, b, the hydraulic depth is equivalent to depth of flow, y, and equation (i) can be rewritten as

(ii)
$$Fr = \frac{q}{y}\frac{1}{\sqrt{gy}}$$

where $q = Q/b$, the flowrate per unit width of the channel. Critical flow in open channels occurs when the Froude number equals unity. The Froude number further characterizes subcritical and supercritical flow as follows:

$Fr > 1$ Supercritical Flow

- Disturbances travel downstream
- Upstream water levels are unaffected by downstream controls

$Fr < 1$ Subcritical Flow

- Disturbances travel upstream and downstream
- Upstream water levels are affected by downstream controls

An equation relating the upstream and downstream depths of a hydraulic jump can be derived from the momentum equation, assuming a uniform velocity profile across the flow area, negligible boundary friction, and a small channel slope. This hydraulic jump equation can be expressed as

(iii)
$$y_2 = \frac{y_1}{2}\left(\sqrt{1+8Fr_1^2}-1\right)$$

Equation (iii) demonstrates that $y_2/y_1 > 1$ only when $Fr_1 > 1$ and $Fr_2 < 1$, thus proving the necessity of supercritical flow for the formation of a hydraulic jump. Another way to visualize this necessity is by using **specific force**, F_s, defined as

(iv)
$$F_s = \frac{q^2}{gy} + \frac{y^2}{2}$$

where the term q^2/gy is the momentum of the flow passing through the channel section per unit time per unit weight of water, and the term $y^2/2$ is the force per unit weight of water. It becomes evident from a plot of F_s as a function of depth for a constant flowrate, or a *specific force diagram*, that the solution to equation (iii) occurs when $F_{s,1} = F_{s,2}$. The depths y_1 and y_2 at which $F_{s,1}$ and $F_{s,2}$ occur are called **sequent depths**.

A stable hydraulic jump will form only if the three independent variables (y_1, y_2, Fr_1) conform to the relationship given in (iii). The upstream depth, y_1, and the Froude Number, Fr_1, are controlled by an upstream headgate for a given discharge. The downstream depth is controlled by a downstream tailgate and *not by the hydraulic jump*. Denoting the actual measured downstream depth as y_2, and the computed sequent depth as y_2', found from (iii), the following observations can be made;

If $y_2 = y_2'$ a stable jump forms;
If $y_2 > y_2'$ the downstream specific force is greater than that at the upstream end, and the jump tends to move upstream;
If $y_2 < y_2'$ the downstream specific force is less than that at the upstream end, and the jump tends to move downstream.

Specific energy, E, in a channel section is defined as the energy per unit weight of water at any section of the channel measured with respect to the channel bottom.

(v)
$$E = y + \frac{V^2}{2g}$$

For a rectangular channel of constant width and constant discharge, equation (v) becomes

(vi)
$$E = y + \frac{1}{2g}\frac{q^2}{y^2}$$

A plot of E vs. y for a constant magnitude of q is called a specific energy curve. The depth of flow at which the specific energy is a minimum for a given discharge is the critical depth, y_c. Applying equation (vi) both at the upstream depth, y_1, and the downstream depth, y_2, the energy loss through the jump may then be evaluated as

(vii)
$$\Delta E = E_1 - E_2$$

The figure below illustrates the relationships between the depths at the upstream and downstream ends of a jump and their corresponding specific force diagram and specific energy curve.

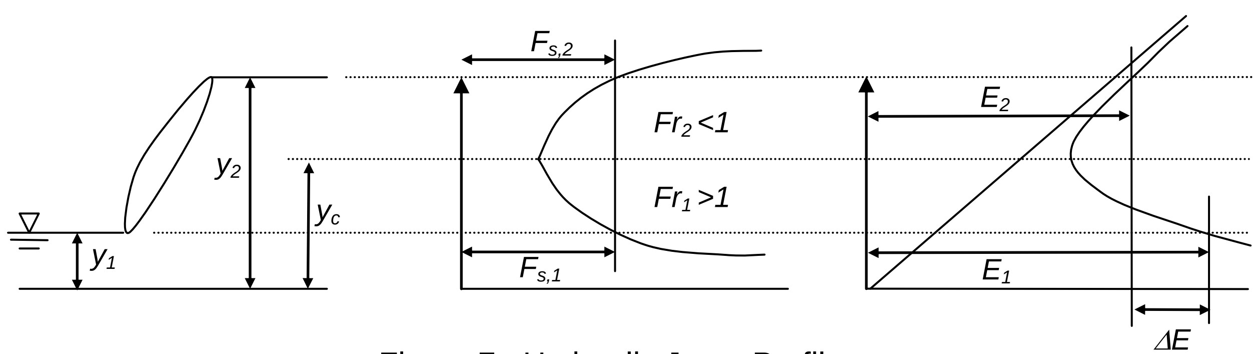

Figure 7: Hydraulic Jump Profile,
Specific Force Diagram, and Specific Energy Curve

II. Objective

The objective is to investigate the validity of the specific force and specific energy equations for the hydraulic jump phenomenon.

III. Procedure

1. Measure the flume width, b , and confirm that the flume is in a horizontal position
2. Start the flow in the flume by turning on the pump.
3. After the system reaches a steady state, record the discharge
4. Position the upstream gate, or headgate, so that the upstream water level is near the top of the flume.
5. Position the tailgate to create a hydraulic jump in the center of the flume
6. After the jump stabilizes, measure and record the depth at a point immediately upstream of the jump and a point downstream of the undulating water surface caused by the jump.
7. Repeat steps 4 – 6 for six headgate positions.

IV. Results

1. For the constant flowrate, compute the critical depth, y_c, using equation (ii) for critical flow conditions.
2. Compute the specific energy and specific force for each depth of each trial using the measured values of supercritical, y_1, and subcritical, y_2, depths.
3. For each depth measured, compute the corresponding Froude number.
4. For each jump, compute the sequent depth, y_2', for each measured value of y_1.
5. What can you infer about the stability of each jump based on the computed sequent depth and the laboratory data?
6. Plot a specific energy curve and a specific force diagram for your laboratory data. Show all pertinent depths (y_1, y_2, y_2', and y_c) and indicate the energy loss, ΔE, for one hydraulic jump.

EXPERIMENT 8
CENTRIFUGAL PUMP CHARACTERISTICS

- *Pump Performance*
- *Characteristics Curve*

I. Introduction

A pump is a device that supplies mechanical energy to fluid flow systems. The general types of pumps in use today are centrifugal, rotary and reciprocating. In hydraulic engineering, the **centrifugal pump** is used almost exclusively. Its advantages include easy operation and repair, low capital cost, and small size required per discharge capacity. The major disadvantages, however, are that it may not be self-priming, it has very limited suction lift, and it has good operating efficiency for only a small range of discharges and heads.

When energy, also called pump head (h_p), is continuously supplied to the fluid system, the energy equation can be written as

(i) $$\frac{p_1}{\gamma}+\frac{V_1^2}{2g}+z_1+h_p=\frac{p_2}{\gamma}+\frac{V_2^2}{2g}+z_2+h_L$$

where z is elevation relative to a datum, V is flow velocity, g is the gravitational constant, p is pressure, γ is the specific weight of the fluid, and h_L is the total energy loss between the pump inlet (1) and the outlet (2). If energy losses are negligible and the diameters of the inlet and outlet pipes are the same, equation (i) can be reduced to

(ii) $$h_p=\frac{p_2}{\gamma}-\frac{p_1}{\gamma}+z_2-z_1$$

The useful power output of the pump, P_{out}, is the power that is consumed by the fluid system in producing a total pump head at a given discharge rate. Mathematically, output can be expressed as

(iii) $$P_{out}=\gamma Q h_p$$

where (is the specific weight of the fluid. The power supplied to the pump, P_{in}, can be evaluated by

(iv) $$P_{in}=V_i A_i$$

where V_i is the electrical voltage and A_i is the electrical current to the motor. Due to mechanical losses in the bearings, fittings, etc., and electro-mechanical losses in the motor, the output power will always be less than that supplied to the pump. The magnitude of these losses are indicated through the efficiency, which can be expressed as

(v) $$e = \frac{P_{out}}{P_{in}} \times 100\%$$

The performance characteristics of a pump are defined as the relationship between operating variables: discharge (Q), head (h_p), power supplied to the pump (P_{in}), and efficiency (e). These characteristics are usually expressed in graphical form in which, for a constant speed, Q (abscissa) is plotted vs. H, P_{in}, and e (ordinates). These curves are then analyzed to select the optimum pump for a particular application.

II. Objective

The objective is to create a pump characteristic curve that describes the performance of a centrifugal pump.

III. Procedure

1. Open the sump drain valve on the bottom of the hydraulics bench and close the control valve on the discharge manifold.
2. Press 'RUN' on the controller keypad to start the pump. Next, press 'FUNC/DATA' until the red LED labeled 'Hz' is illuminated and use the arrow keys to adjust the motor speed to 50 Hz. Fully open the discharge control valve.
3. Determine the flowrate by closing the ball valve in the volumetric tank and measuring the time required to accumulate a known volume of fluid in the tank. Use a digital timer and the sight glass located on the side of the hydraulic bench.
4. Record the suction (inlet) head and discharge (outlet) head from the pressure gauges. Using the 'FUNC/DATA' key to select the appropriate reading, record the pump input voltage and current from the digital controller. Assuming a datum through the center of the pump impeller, the distance from the datum to the inlet gauge is 0.020 m (z_1) and from the datum to the outlet gauge is 0.170 m (z_2) .
5. Repeat steps (3) and (4) for at least 5 flowrates. For each trial, you will need to release the ball valve to drain the volumetric tank and close the discharge valve slightly to obtain a new discharge.
6. Repeat steps (3) – (5) for pump speeds of 40 and 30 Hz.

IV. Results

1. Plot on *one graph* the pump performance curves for the three motor speeds. For each speed, plot discharge in m^3/s (abscissa) vs. (1) total head delivered by the pump in meters from equation (ii); (2) power supplied to the pump in watts from equation (iv); and (3) efficiency in percent from equation (v). As an example only, the figure below illustrates the expected results for a different pump.

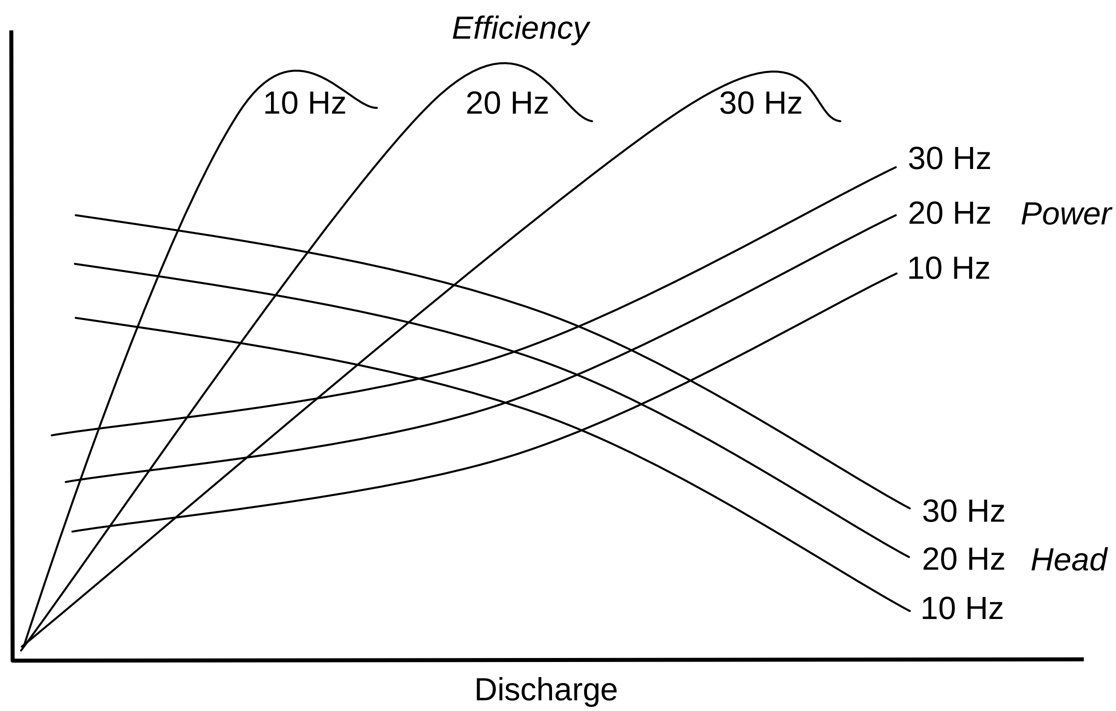

Figure 8: Pump Characteristic Curves

EXPERIMENT 9
MOBILE BED FLOW VISUALIZATION

- *Flow Patterns*
- *Drag Force*
- *Sediment Scour and Deposition*

I. Introduction

The Ahlborn tank, named after a German engineer in the early 1900's, is particularly useful for demonstrating important fluid phenomena in engineering practice. The tank consists of slow, shallow flow of water at a constant depth over a horizontal channel bed. Flow around structures and objects can be easily investigated by installing model devices in the channel. For example, a circular cylinder in the Ahlborn tank can be used to simulate flow patterns around a circular bridge pier, as illustrated in the figure below.

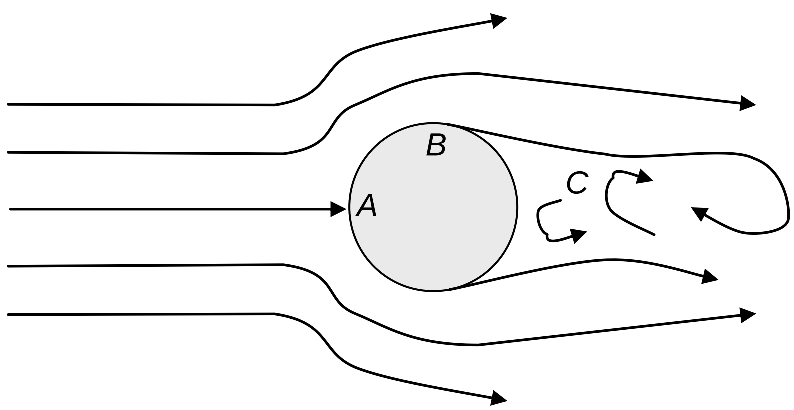

Figure 9: Flow Around a Circular Cylinder

The cylinder experiment allows clear observation of the stagnation point at *A* on the central streamline; acceleration of flow from *A* to *B*; flow separation at *B*; a turbulent wake at *C* where eddies are alternately generated on opposite sides to give a sinuous pattern farther downstream; and diminishing curvature of streamlines at locations further downstream of the cylinder. The friction induced by the cylinder's boundary is referred to as **skin drag** and occurs primarily within the **boundary layer**. This is a very thin layer at the object's surface. A secondary component of the total drag force acting on the cylinder is **form drag**, which results from the large pressure differences between the upstream and downstream ends of the object. The latter is primarily responsible for the formation of the turbulent wake at the downstream end.

Additional important phenomena that can be modeled in an Ahlborn tank are sediment scour (e.g. erosion) and deposition. Alluvial rivers and reservoirs continually change their position and shape as a result of the hydraulic forces acting on their beds and banks. Specifically, movement of sediment occurs when shear stresses reach a critical level, called the **critical shear stress**, which defines the **threshold of motion** for particles. Changes to rivers and reservoirs may be slow or rapid and can result from either natural environmental stimuli or human activities. When a river is locally modified in some way, such as construction of bridge piers or creation of stream diversions, corresponding changes in channel characteristics will often propagate both upstream and downstream, sometimes for long distances. The impact can be significant; consider that deposition in rivers and reservoirs reduces storage capacity and interferes with navigation, and erosion can undermine bridge piers and underground utilities.

For sedimentation processes modeled in the Ahlborn tank, the timescales required for equilibrium scour depths to be achieved are incorrect, as much longer periods are required in reality. However, experience has shown that if erosion and deposition is allowed to proceed in the tank until equilibrium is reached, the final patterns are approximately to scale. In addition, since there is no provision for feeding in sediment at the upstream end and the tank is short, experiments involving regional movement of sediment cannot be accurately modeled.

II. Objective

The objective is to investigate flow patterns and sediment movement around various simulated hydraulic structures and devices in alluvial rivers.

III. Procedure

1. Check to ensure that the sump tank of the Ahlborn tank is filled with water to a depth of approximately 80 cm. Turn on the pump and bleed any air from the piping system.
2. Slowly open the control valve, which directly increases flowrate, until a suitable discharge is obtained. Discharge is read from the digital display on the control panel. A suitable flow should result in an equilibrium depth of approximately four to six centimeters in the channel.
3. Beginning with a plane bed, simulate a circular bridge pier or pile by inserting a cylinder on the centerline of the tank approximately 0.5 m from the upstream end. Convergent plates can be used in the channel to exaggerate the sedimentation effects. Once the sediment has reached an approximately stable condition, close the control valve and analyze the bed sediment response. Measure the depths of scour and deposition at the upstream and downstream ends of the pier. Smooth the surface of the channel bed prior to conducting additional experiments.
4. Repeat the experiment using a rectangular pier (a) placed with the shortest edge perpendicular to the flow and (b) with its centerline at approximately 30° from the flow direction.

5. Repeat the experiment using both sharp-nosed and rounded-nose piers placed parallel to the flow direction.
6. Using a similar procedure, simulate a meandering river by installing the two curved channel plates. Using white paint, analyze the flow pattern through the meandering channel.
7. Again using a similar procedure, insert the model river bend in the channel to simulate flow in a sharp bend. Sediment does not need to line the bed of the bend. Use white paint to evaluate the velocity and depth distribution at a particular cross section in the bend.
8. Finally, insert the airfoil approximately 0.5 m from the upstream end of the channel. Using white paint to visualize flow patterns, analyze the change in form drag, evidenced by the turbulent wake and sediment movement, as a result of increasing/decreasing the airfoil's angle of attack.

IV. Results

1. Sketch and compare the bed sediment response, both scour and deposition, on the various simulated bridge piers. Which pier experiences the least amount of sediment scour and deposition? Which pier would be most economical?
2. Sketch and describe the flow pattern and bed sediment response through the simulated meandering river. What effect does the channel width have on sediment movement and why? Hypothesize what will happen to a meandering river such as the one modeled over time; will the meanders or curvatures grow, or will the river begin to straighten naturally?
3. For a cross section of the model river bend, sketch a profile of depth of flow and label areas of higher and lower velocities.
4. Describe the general relationship between the airfoil's angle of attack and form drag. Where are the regions of highest and lowest pressure around the airfoil? Can you explain a qualitative relationship between the magnitude of form drag and the extent of sediment movement around the airfoil?

EXPERIMENT 10
MINOR LOSS EXPERIMENT

- *Loss Coefficient*
- *Minor Energy Losses*

I. Introduction

The losses of energy, or head, in full-flowing conduits can be classified into two components: (1) energy loss due to the frictional resistance of the conduit walls to flow, and (2) energy loss due to the pipe fittings and appurtenances (e.g., bends, contractions, and valves). The latter is referred to as minor, or form, loss and is associated with a change in magnitude and/or direction of the flow velocity. Generally, the more abrupt the change, the higher the associated energy loss.

For a long pipeline (*L/D* > 2000), the energy loss is predominantly associated with friction and minor losses are small. However, minor losses would comprise a considerable part of the total energy loss for a system that is relatively short and has a large number of fittings. Therefore, it is important for a designer to carefully consider both types of losses in the design of distribution systems.

To determine the head loss across a pipe appurtenance, consider the energy equation written between two sections: immediately before (1), and after (2) the pipe appurtenance,

(i) $$\frac{p_1}{\gamma} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{\gamma} + \frac{V_2^2}{2g} + z_2 + h_l$$

where z is the elevation of the centerline of the pipe relative to an arbitrary datum, V is flow velocity, g is the gravitational constant, p is pressure at the centerline of the pipe, γ is the specific gravity of the fluid, and h_l is the head loss between sections *1* and *2*. When only a short distance separates sections *1* and *2,* h_l is a direct measure of minor loss. The velocities in equation (i) can be evaluated if the flowrate and pipe dimensions are known. If the pressure at sections *1* and *2* can be measured, the energy equation can then be used to evaluate the unknown head loss through the pipe.

The energy loss that occurs through a pipe fitting, is commonly expressed in terms of velocity head in the form

(ii) $$h_l = K\frac{V^2}{2g}$$

where K is the dimensionless minor loss coefficient for the pipe fitting, and V is the mean velocity of flow into the fitting.

Because of the complexity of flow through various fittings, *K* is usually determined by experiment. In this case, the head loss is calculated from two manometer readings, taken before and after each fitting, and *K* is then determined as

(iii)
$$K = \frac{\Delta h}{\frac{V^2}{2g}}$$

For contractions and expansions, an additional change in static pressure is experienced due to the change in pipe cross-sectional area through the enlargement and contraction. This change can be calculated as

(iv)
$$\frac{V_1^2}{2g} - \frac{V_2^2}{2g}$$

where V_1 and V_2 are the upstream and downstream velocities respectively. To eliminate the effects of this area change on the measured head losses, this value should be added to the head loss reading for an enlargement, and subtracted from the head loss reading for a contraction.

For a gate valve, pressure difference before and after a valve can be measured directly using a pressure gauge. This can be converted to an equivalent head loss using the equation

(v)
$$1\, bar = 10.2\ m\ water$$

II. Objective

The objective is to determine the loss coefficients for flow through a range of pipe fittings including bends, a contraction, an enlargement, and a gate valve.

III. Procedure

1. Open the bench valve, the gate valve and the flow control valve and start the pump to fill the test rig with water.
2. Bleed air, if present, from the pressure tap points and the manometers by adjusting the bench and flow control valves and air bleed screw.
3. Check that all the manometer levels lie within the scale when all the valves are fully opened. Adjust the levels, if necessary, using the air bleed screw and the hand pump.
4. For a selected flow rate, record the reading from all the manometers (that are tapped before and after each appurtenance: enlargement, contraction, long bend, short bend, elbow, miter) after the water levels have steadied.

5. Determine the flow rate by accumulating a fix volume of water in the volumetric tank with help of a stopper. Use a digital stopwatch to record time and the sight window of the bench to find the volume of water.
6. Repeat steps (4) and (5) for two more flow rates.
7. Clamp off the connecting tubes to the miter bend pressure tappings (to prevent air from being drawn into the system). Start with the gate valve fully closed and the bench valve and control valve fully open. Open the gate valve 50% of its total opening (after taking up any backlash). Record the gauge reading for the half open condition.
8. Adjust the flow rate with the control valve and measure pressure drop across the gate valve from the pressure gauge. Also, measure the volume flow rate by timed collection of water.
9. Repeat the step (8) for two more flow rates.

IV. Results

1. Calculate head loss (h_l) across the fittings for each flow rate in step (4) - (6).
2. Calculate the velocity head for each flow rate. Then calculate K for each bend using equation (iii); for the contraction and the enlargement using equations (iii) and (iv); and for the gate valve using equations (iii) and (v).
3. For each pipe fitting, plot head loss (h_l) vs. $V^2/2g$, and K vs. volumetric flow rate, Q.
4. Discuss your results. Specifically, comment on whether it is justifiable to treat the loss coefficient as a constant for a given fitting.

References

Chadwick, A. and J. Morfett, 1986, *Hydraulics in Civil Engineering*, Allen & Unwin, Ltd., London.

Chevalier, L.R. and Craddock, J.N., 2002, *NSF Lab Project: Civil and Environmental Engineering Laboratories*, Available Online: < http://civil.engr.siu.edu/nsflab/ >.

Chow, V.T., 1959, *Open-Channel Hydraulics,* McGraw-Hill, New York, NY.

Hwang, N.H.C, 1981, *Fundamentals of Hydraulic Engineering Systems,* Prentice-Hall, Inc., Englewood Cliffs, NJ.

Mott, R.L., 1990, *Applied Fluid Mechanics*, 3^{rd} Ed., Merrill Publishing Co., Columbus, OH.

Prasuhn, A.L., 1980, *Fundamentals of Fluid Mechanics*, Prentice-Hall, Inc., Englewood Cliffs, NJ.

Crowe, C.T., J.A. Roberson and D.F. Elger, 2001, *Engineering Fluid Mechanics*, 7^{th} Ed., John Wiley & Sons, Inc., New York, NY.

Ruff, P.F., 1990, *Experiments in Hydraulic Engineering*, Department of Civil Engineering, Arizona State University, Tempe, AZ

Street, R.L., G.Z. Watters and J. K. Vennard, 1996, *Elementary Fluid Mechanics*, 7^{th} Ed., John Wiley & Sons, Inc., New York, NY.