KENNESAW STATE UNIVERSITY

Southern Polytechnic College of Engineering and Engineering Technology

# Fluid Mechanics Laboratory Manual

## ENGR 3345 – Fall 2020

David S. Ancalle, P.E.
Aijia Xu

Fluid Mechanics Laboratory Manual
written by David S. Ancalle
last updated: 8/14/2020

## Contents

Preface ........ 2
Course Outline ........ 2
Report Expectations ........ 3
Viscosity ........ 4
Fluid Statics ........ 11
Free Jets ........ 17
Pelton Wheel Turbine ........ 22
Closed Conduit Flow ........ 27
Pump Performance ........ 34
Open Channel Flow ........ 39

## Preface

This document contains the lab manual for the course ENGR 3345: Fluid Mechanics Laboratory. The experiments in the laboratory were developed over the years by various faculty at the former Southern Polytechnic State University. In recent years, the materials were revised by Prof. M. A. Karim and Prof. Tien M. Yee. The current version of the manual was written by me for exclusive use within my own lab sections. Other instructors are welcomed to use this manual (just let me know first). I am indebted to the original work done by all the faculty members that came before me. Special thanks should go to Prof. Huidae Cho (University of North Georgia), Prof. Tien M. Yee (KSU) for their help in putting together this material. To the student, I am always looking to improve on my work, so I am open to feedback on how to improve this manual.

David S. Ancalle
Kennesaw State University

## Course Outline

This laboratory course reinforces the concepts learned in *ENGR 3343: Fluid Mechanics* through practical experiments where students can measure, analyze, and present data. Students should read the section for each experiment, watch the corresponding video, and complete a short quiz prior to coming to the classroom. On each meeting day, students will then measure data for their experiment, which will take between 45 minutes and 1 hour. After the data is recorded, students may leave or remain in the lab to go over any remaining questions before submitting the report.

Since students are expected to learn the course contents in the Fluid Mechanics lecture, the focus of this laboratory will be in preparing technically sound and professional reports. Each report will be evaluated thoroughly and rigorously. Reports will be assessed on the clarity of their presentation, the accuracy of the results, and the quality of the discussion, analysis, & conclusion.

As students may have different goals on whether to move on to industry or academia after graduation, this laboratory will emphasize three types of reports: an *engineering report*, a *research article*, and a *poster presentation*. Of the seven reports to turn in, three must be written as engineering reports and three must be written as research articles. One report (Lab 6) will be written as a poster presentation.

## Report Expectations

Engineering reports should have the following sections: executive summary, project narrative, results, discussion, analysis, conclusion, and appendix. The executive summary is a one-paragraph summary of the entire report. It should state the objectives of the experiment, the general findings, and the conclusions. The project narrative should introduce the topic and explain the theory. It is not necessary to be mathematically rigorous in this section, but rather, it should focus on practical applications. The conclusions should focus on whether the objectives of the lab were met. In lieu of a references section, the appendix should include any reference data such as graphs, tables, and other information used from external sources, as well as detailed calculations when applicable. Normally, figures are colored and may be included in the body of the report or the appendix. Tables are usually included in the body of the report. For reference, see the provided sample report, which shows the minimum acceptable standard by which your reports will be graded.

Research articles should have the following sections: abstract, introduction, results, discussion, analysis, conclusion, and references. The abstract is a one-paragraph summary of the objectives, findings, and conclusions of the article. The introduction should provide a mathematically rigorous theory behind the topics. Practical applications are not emphasized, but reference must be made to previous work in the topic. The conclusion should focus on whether the objectives of the lab were met. The references section should point to any outside source used in the making of the article. It is preferable to use primary references where applicable. Figures should be in black and white and should preferably have as few shades as possible. Figures and tables are included in the body of the report. For reference, see the provided sample article, which shows the minimum acceptable standards by which your articles will be graded.

The poster presentation (Lab 6) is a one-page poster that includes a summarized background, conclusions, and references. Figures and graphics are emphasized and the main form of communication will be visual, so all figures should be well labeled and identified. The poster should have a two or three column format. For reference, see the provided sample poster, which shows the minimum acceptable standards by which your poster will be graded.

# Viscosity

## Objective

The objectives of this experiment are: (1) to derive an equation to determine the viscosity of a fluid using the Thomas Stormer viscometer, (2) to develop a methodology to calibrate viscosity measurements from a Cole Palmer viscometer, and (3) to recommend the best viscometer for use in measuring the viscosity of oils.

## Theory

Viscosity can be informally defined as a measure of the internal friction of a fluid. When exposed to certain types and magnitudes of stress, matter will tend to deform. For fluids, the resistance to such deformation from stress is known as viscosity. Think of a fluid flowing through a pipe. There is friction between the walls of the pipe and the "edges" of the fluid (i.e. the part of the fluid that is in contact with the pipe walls). The pipe wall will exert a shear stress that will slow the fluid down. As you move away from the pipe wall and towards the center, the shear stress will decrease and the velocity of the fluid will increase.

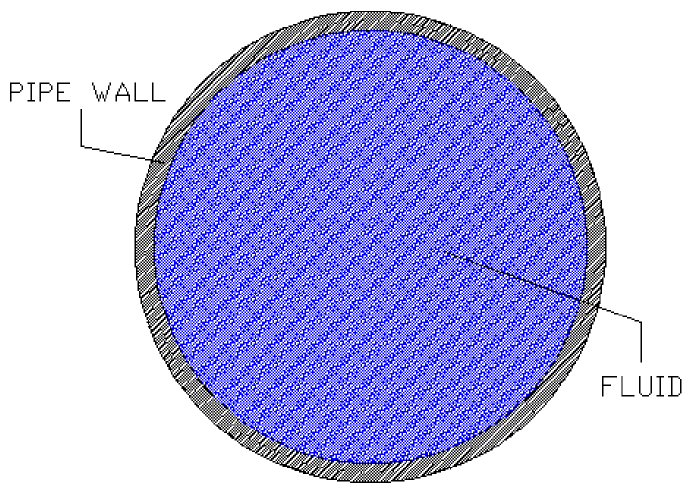



Figure 1.1 Cross sectional view of a full pipe

Even though the friction of the pipe wall is directly acting only on the surface of the fluid that is in contact with the pipe, there is still some indirect effect of that friction along the rest of the fluid. As the pipe wall exerts friction on the fluid particles in the edges, those fluid particles will exert their own "internal friction" on the other fluid particles in contact with it, and so on. This in turn results in a velocity and a shear stress gradient on the fluid. The amount of friction force that will be exerted on the fluid at any point in reaction to the friction from the pipe wall is proportional to the fluid's viscosity.

An easier way to understand viscosity is to look at the difference between syrup and water. Syrup has a higher viscosity than water; therefore, it will take longer to pour a given volume of syrup than it would to pour the same volume of water. The higher viscosity of the syrup means that its particles have more resistance to deformation between them, so, when pouring syrup (and exposing it to gravitational force), the syrup will have a higher resistance to deformation than the water.

Going back to the example of pipe flow, if we were to trace a line between the pipe wall and the center of the pipe (called the y-axis), and measured the velocity at various points along the line (called the v-axis), we would see that the closer the line is to the pipe wall, the smaller the velocity is. In a no-slip condition, the velocity at the point directly on top of the pipe wall would be zero, and the maximum velocity would be obtained at the point furthest from the pipe wall (the center).

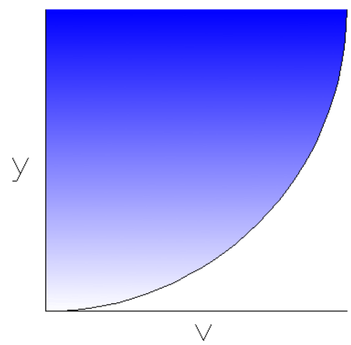


Figure 1.2 Velocity gradient from the pipe wall to the center

The relationship between the velocity of the fluid at a certain point and the distance between that point and the pipe wall is known as a velocity gradient, $du/dy$. Since we know that fluid in motion inside a pipe will be subject to a shear stress, $\tau$, and that the shear stress will decrease as the fluid is further away from the pipe wall, then we know that the shear stress is in some way related to the velocity gradient. Isaac Newton found that this relationship is linear for several fluids (which we now call Newtonian fluids). The factor by which the shear stress is proportional to the velocity gradient is what we call viscosity, $\mu$.

$$\tau = \mu \frac{du}{dy} \tag{1.1}$$

In this experiment, the viscosity of SAE30 motor oil will be measured with a viscometer. A viscometer is an instrument that places a drum in contact with a fluid. A torque is then applied to the drum to make it rotate inside the fluid. The resistant force of the fluid to the shear stress caused by the drum can be determined from measurements of the drum's angular velocity and the torque being applied to the drum. If we assume a linear velocity profile, then the velocity gradient of the fluid is:

$$\frac{du}{dy} = \frac{\omega r_i}{Y} \tag{1.2}$$

where $\omega$ is the angular velocity of the drum; $r_i$ is the drum's radius; and $Y$ is the thickness of the fluid film. The torque applied to the fluid can be determined from the geometry of the drum, including its height $h$:

$$T = 2\pi\tau h r_i^2 \tag{1.3}$$

The first viscometer to be used in this lab is the Thomas Stormer viscometer. This instrument consists of an object that is attached via a string to the drum. If the object is dropped, it will make the drum rotate. Assuming no friction, we can say that the power generated from the free-falling object is equal to the power applied to the drum. From physics, we can determine the power generated by the free fall:

$$\dot{W} = \vec{F}_W \cdot \vec{\mathrm{v}} = \frac{F_W L}{t} \tag{1.4}$$

where $F_W$ is the object's weight; $L$ is the distance it falls; and $t$ is the time it takes to fall that distance. This equation assumes constant velocity. The falling object will make the drum rotate $n$ times. Thus, the angular velocity of the drum is:

$$\omega = \frac{2\pi n}{t} \tag{1.5}$$

Using this information, we can relate the object's falling time to the fluid's viscosity, $\mu = f(t)$.

The second instrument that will be used to measure viscosity is the Cole Palmer viscometer. This machine performs the computations electronically. The Cole Palmer viscometer rotates a spindle submerged inside a fluid at a given velocity and measures the amount of power required to rotate the spindle. It then uses that power measurement to compute the viscosity of the fluid. The Cole Palmer viscometer was designed to measure the viscosity of very thick fluids, such as paint, therefore it is not calibrated to use with SAE 30 motor oil. A calibration method must be employed to adjust the data taken from the Cole Palmer viscometer.

**Procedure**

Thomas Stormer viscometer

1. Based on the theory, determine which properties of the Thomas Stormer viscometer are relevant to the calculation of viscosity and measure these properties.

2. Place the Thomas Stormer viscometer at the edge of the table to allow for the $50\ g$ object to drop freely towards the ground to test the viscometer.

3. Fill the outer cylinder with SAE 30 oil. Place the outer cylinder back into the platform, but make sure that the notch at the platform locks the groove of the outer cylinder.

4. Raise the platform to ensure that the drum is fully submerged in SAE 30 oil. If not, add the oil until fully submerged, but be very careful to not overfill. Also, make sure that the drum is not directly in contact with the outer cylinder at the sides and bottom. Test the apparatus once.

5. Once ready, the gage of the Thomas Stormer reads the number of rotations of the drum. Release the break to allow the weight to fall freely and observe and record the time it takes for the drum to rotate 100 times.

6. Repeat to obtain a total of 5 time measurements.

Cole Palmer viscometer

1. Ensure that the viscometer is level.

2. Turn the viscometer on. Follow the instructions on the screen to auto-zero and calibrate the viscometer.

3. Attach the #2 spindle.

4. Fill the beaker with the SAE 30 oil at room temperature.

5. Lower the spindle into the beaker until the spindle groove is at the liquid level and at the center of the beaker.

6. Set the spindle speed to 60 RPM.

7. Begin measuring. As the spindle is spinning, ensure that there are no air bubbles attached to it. Also check the bubble level on the apparatus to ensure that it is still level.

8. Allow for sufficient time for the displayed measurements to stabilize. Then start recording measurements.

9. Increase the temperature by $5\ ℃$ and obtain the viscosity and temperature reading. Be sure to wear gloves when handling the heating plate and beaker.

**Results**

1. Using the geometric measurements of the Thomas Stormer viscometer, formulate an equation that relates the time it takes for the object to drop to the viscosity of the fluid. Show all your work.

2. Create a table for the Thomas Stormer viscometer measurements. This table should include the measured time to 100 revolutions (in seconds), and the calculated viscosity (in $Pa \cdot s$) of the fluid using the equation you formulated.

3. Find a value of viscosity for SAE30 oil under room temperature from published literature and compare it to your viscosity measurement.

4. Find values of viscosity for SAE30 oil for varying temperatures to be used in the next step.

5. Create a table for the Cole Palmer viscometer measurements. This table should include the measured temperature, measured viscosity, published viscosity, and percent error between the measured and published values.

6. Create a graph of percent error vs. temperature. Fit a curve to this data.

7. Calibrate the data from your Cole Palmer viscosity measurements to the published values and document the steps taken for the calibration. The simplest way to come up with a form of calibration is to observe errors between your data and published data vs. temperature and fit a regression curve to describe the

errors (use your graph of error vs. temperature). Try varying the polynomial order of your regression curve and select the most appropriate polynomial order for your calibration curve. Once a trend line is selected, future collected data using the Cole Palmer viscometer can be adjusted using your regression equation.

8. Create a table that shows the temperature and the calibrated viscosity.

9. Create a graph of calibrated viscosity vs. temperature.

**Discussion**

1. Compare the viscosity obtained from the Thomas Stormer viscometer to the published value. If there is a major difference, suggest factors that may have contributed to the difference in values.

2. Explain how the time obtained from the Thomas Stormer viscometer would have varied if we had used objects with different mass. Give examples of approximate time values you would have obtained had you used objects of different masses.

3. Explain the steps taken to develop a calibration curve for the Brookfield viscometer. Also provide a justification for your choice of the polynomial order used on your calibration equation.

4. Comment on how the viscosity of oil changes with temperature and explain why this may happen.

5. Compare the consistency and accuracy of the Thomas Stormer viscometer and Cole Palmer viscometers.

**Analysis**

1. What are the advantages and disadvantages of using the Thomas Stormer viscometer? Consider the following in your response: accessibility of equipment, consistency/accuracy of results, ease of usage, and potential for human error.

2. Do you think the equation you derived for the Thomas Stormer viscometer may be used to measure the viscosity of other fluids? Why or why not?

3. What are the advantages and disadvantages of using the Cole Palmer viscometer?

4. In order to develop the calibration curve, what assumptions are being made regarding the relationship between Error and temperature? Do you think this calibration curve can work for fluids other than SAE 30 oil? Why or why not?

5. Identify factors that could have led to the corruption of data in this experiment. Some examples include: human error, integrity of the fluid, & condition of the equipment. Also identify the assumptions being made in this experiment that can compromise the data or the way we compute our results.

**Conclusion**

1. Present your equation for a Thomas Stormer viscometer and state its uses and limitations.

2. Present your calibration equation for the Cole Palmer viscometer and state its uses and limitations.

3. Recommend the best viscometer for measuring the viscosity of oil.

**Sample Data Sheet**

Experiment 1: Viscosity

Thomas Stormer Viscometer

| | |
|---|---|
| Drum radius ($mm$) | 19 |
| Drum height ($mm$) | 38.1 |
| Oil thickness ($mm$) | 5 |
| Object mass ($g$) | 50 |
| Drop length ($mm$) | 500 |

| Trial | Time to 100 revolutions ($s$) |
|---|---|
| 1 | 116 |
| 2 | 113 |
| 3 | 112 |
| 4 | 115 |
| 5 | 116 |

Cole Palmer Viscometer

| Temperature (℃) | Viscosity ($mPa \cdot s$) |
|---|---|
| 23 | 8.25 |
| 29 | 7.81 |
| 34 | 7.00 |
| 39 | 6.63 |
| 44 | 5.50 |
| 49 | 3.23 |
| 54 | 3.00 |
| 59 | 2.88 |
| 64 | 2.75 |
| 69 | 2.06 |

# Fluid Statics

**Objective**

The objectives of this experiment are: (1) to recommend the best method for measuring hydrostatic pressure using an Edibon quad, (2) to calculate the specific weight of water using Archimedes' principle.

**Theory**

In a static system, the sum of all forces and moments at any point and in any direction should equal zero. This does not mean that there are no forces acting on a static system, but rather, that when combined, the forces in the system will cancel out and produce no acceleration. Fluid statics is the study of the internal and external forces acting on a fluid that has no net acceleration.

If we take a static reservoir (an open container with water that is not moving), the forces acting on this system are many. To name a few: (1) the weight of the fluid, (2) the resulting normal force on the bottom of the reservoir, and (3) the resultant force from hydrostatic pressure on the sides of the reservoir. If we were to insert an object into the reservoir and look at the forces acting on that object, we would have (4) the weight of the object, (5) the weight of the water directly on top of the object, (6) the resultant force from hydrostatic pressure on the sides of the object, and (7) the buoyant force from the water acting on the object. These forces will be taken into account when deriving our equations for this experiment.

Considering a hydrostatic system consisting of a reservoir filled with water; let's take a differential volume of water (a small "cube" of water with dimensions dx, dy, and dz). The forces acting on this differential cube include: (1) the weight of the column of water directly above the cube (2) the normal or "buoyant force" of water acting on the bottom of the cube (we'll discuss buoyancy in detail during the experiment)

From a theoretical perspective, these forces acting on the z axis of the cube will cause it to compress vertically, and therefore, expand horizontally. This expansion will result in a force between the sides of the cube and the water surfaces it is interacting with, equal to the magnitude of the forces acting on the z axis of the cube. The distribution of these forces over the surface area of the cube is known as hydrostatic pressure. Let's begin by taking into account the top side of the cube (since this is a static system, the forces acting on every side of the cube are equal to each other). The force acting on the top side of the cube is the weight of the "water column" directly on top of the cube. This weight can be expressed as: $F_W = mg$. The area that the weight of the column is acting on the cube can be defined as $dA = dx\ dy$. Therefore, the pressure on the top face of the cube is: $P = \frac{F}{A} = \frac{F_W}{dA} = \frac{mg}{dx\ dy}$. The density of the fluid in the reservoir is equal to the mass of the fluid over the volume it occupies. The density of the water column on top of the cube is then: $\rho = \frac{m}{V} = \frac{m}{dx\ dy\ h}$ where $h$ is the total height of the water column above the cube. This is also equal to the distance from the cube to the surface of the water. We can rearrange this equation and insert it into the pressure equation to obtain:

$$P = \gamma h \tag{2.1}$$

where the specific weight of the fluid is equal to its density times gravitational acceleration. Looking at this equation, we can see that the hydrostatic pressure does not depend on the geometric properties of the "cube" but rather, it depends on the properties of the fluid surrounding it. The only parameter of the "cube" that affects hydrostatic pressure is its depth (distance to the surface of the water). This means that we can use this equation to determine the hydrostatic pressure acting on any object submerged in a fluid, given that we know the fluid's density or specific gravity.

**Procedure**

This experiment will consist of two different activities. The first of these involves measuring the force resulting from hydrostatic pressure using a hydrostatic pressure apparatus. The equipment consists of a water tank with a quadrant and a beam attached to the top. The measurements of the quadrant are shown in Figure 2.1. The second activity consists of using a graduated cylinder, a small object, and a spring weight.

To measure hydrostatic pressure:

1. Place the device on a flat surface close to the sink. Check the bubble level attached to the apparatus and make sure that it is level.

2. Hang the empty weight tray on the left arm of the beam and adjust the counter weight on the right arm to make the beam balanced. Use the bubble level on the beam to make sure that it is in equilibrium.

3. Start filling the tank with water until the water level is approximately at the 30mm mark on the quadrant. As water is added into the reservoir, the beam will be off-balanced again.

4. Measure and record the temperature of the water.

5. Add weights to the weight tray on the left arm of the beam until it is once again balanced.

6. Record the water level and the total mass used to equalize the beam.

7. Repeat this experiment to obtain at least 5 data points by increasing the water level by approx. 10 mm each time.

To measure buoyancy:

1. Fill the graduated cylinder with 900 mL of water.

2. Measure the temperature of water.

3. Hang the object on the spring scale and record its dry weight.

4. Submerge the object into the water in the graduated cylinder.

5. Record the weight reading on the spring scale and the water level in the graduated cylinder when the object is submerged.

6. Repeat this process. Try submerging the object at different depths and note how this affects the weight on the spring scale.

**Results**

Hydrostatic Pressure

1. Find the average pressure on the vertical surface of the quadrant and calculate the theoretical magnitude of the resultant force, $F_{th}$ for each measurement. For this step, you only need to take into account the submerged depth of the surface.

2. Create a table and include the depth of the water ($mm$) from your laboratory measurements. Add columns with the following data: depth of the water converted to meters, submerged area (in square meters), theoretical pressure at the bottom of the surface (in Pascal), average pressure on the vertical surface (in Pascal), and theoretical hydrostatic force on the surface (in Newton).

3. Find the theoretical center of pressure (the point where the resultant force acts on) for each measurement using the moment of inertia around the centroid:

$$y_{th} = \bar{y} + \frac{\bar{I}_x}{A\bar{y}} \tag{2.2}$$

4. Create a table with the water depth (in meters) and the theoretical center of pressure (in meters measured from the bottom of the surface).

5. Use the theoretical center of pressure ($y_{th}$) and the data from the mass of the weights to compute the experimental resultant force,$F_{ex}$. Remember that for the beam to be balanced, the moment around point C in the instrument should be zero.

6. Create a table with the following columns: mass recorded from the laboratory experiments (in grams), mass converted to kilograms, weight of the mass (in Newtons), moment from the mass acting on point C (in Newton-meters), moment from the hydrostatic force (in Newton-meters), distance from the theoretical center of pressure to point C (in meters), experimental force (in Newtons).

7. Use the theoretical magnitude of the resultant force, $F_{th}$ to compute the experimental center of pressure, $y_{ex}$ by setting the moment around point C in the instrument to zero.

8. Create a table with the following columns: mass recorded from the laboratory experiment (in kilograms), weight of the mass (in Newtons), moment from the mass acting on point C (in Newton-meters), moment from the hydrostatic force (in Newton-meters), experimental distance from the center of pressure to Point C (in meters), experimental center of pressure measured from the bottom of the surface (in meters).

Buoyancy

9. Use your weight measurements to calculate the buoyant force acting on the object for each measurement.

10. Use this buoyant force to calculate the specific weight of water.

11. Compare this specific weight to a published value. Report the error of the measured buoyant force.

12. Create a table with the following columns: measured volume (in mL), measured weight of the object (in N), buoyant force (in N), experimental specific weight, published specific weight, and percent error.

**Discussion**

Hydrostatic Pressure

1. Compare your experimental values to your theoretical values. Comment on the differences in values and give possible reasons to explain the discrepancies.

Buoyancy

2. Does the buoyant force on a fully submerged object change with depth? Explain why this does or does not happen conceptually (i.e. using the theoretical definition of buoyant force) and mathematically (i.e. using the basic equations of buoyant force)

3. Give possible reasons to explain the differences between measured and published specific weight.

**Analysis**

Hydrostatic Pressure

1. Explain why we cannot obtain both experimental values at the same time.

2. To the best of your judgment, explain which of these methods is better: to compute an experimental force using the theoretical center of pressure, or to compute an experimental center of pressure using the theoretical force.

3. From this experiment, we can conclude that, if the water level is less than 100 mm, the center of pressure is located at one third of the submerged depth from the bottom of the quadrant. Explain why this is not valid for depths greater than 100 mm.

Buoyancy

4. Will the magnitude of the buoyant force be any different if the object were made of a denser material with the same volume and shape? Explain.

5. Assuming the same density and volume, will the shape of the object affect the magnitude of the buoyant force when the object is fully submerged? Why or why not?

6. Can this methodology be used for other fluids?

**Conclusion**

1. Recommend the best method for calculating hydrostatic pressure with the Edibon instrument (using the theoretical force or theoretical center of pressure).

2. Recommend a methodology to calculate the specific weight of a fluid using only a spring weight and a graduated cylinder.

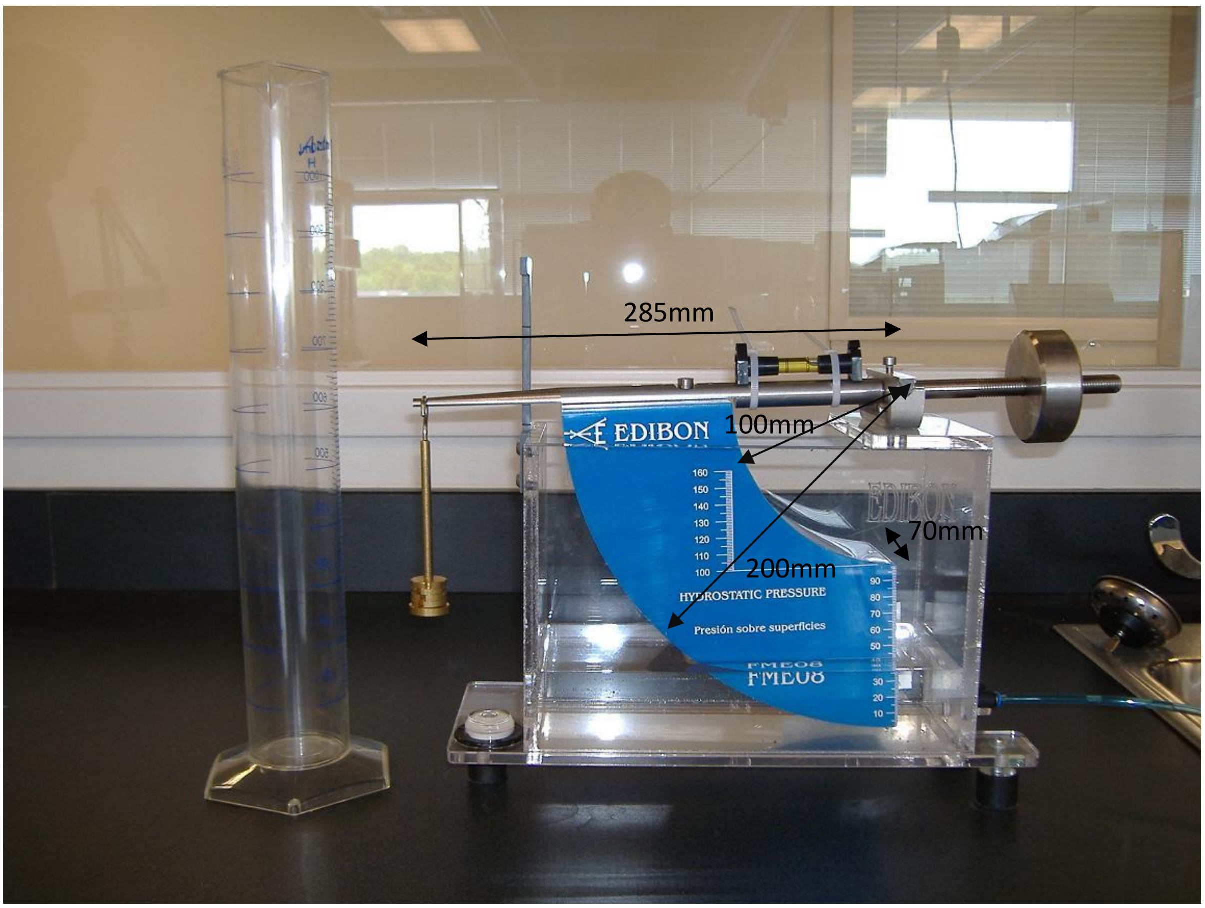


Figure 2.1 Equipment for Fluid Statics experiment (from Prof. Tien M. Yee's lab manual)

Width of surface = 70 mm

**Sample Data Sheet**

Experiment 2: Fluid Statics

Hydrostatic Pressure

Water temperature: 23 °C

| Depth ($mm$) | Mass ($g$) |
|---|---|
| 21 | 10 |
| 30 | 20 |
| 36 | 30 |
| 41 | 40 |
| 46 | 50 |

Buoyancy

Water temperature: 21 °C

Object dry weight: 2.85 $N$

Initial volume: 910 $mL$

| Volume ($mL$) | Object Weight ($N$) |
|---|---|
| 955 | 2.45 |
| 950 | 2.55 |
| 950 | 2.50 |

# Free Jets

**Objective**

The objective of this experiment is to formulate an empirical relationship between the angle of deflection and impact force.

**Theory**

A jet is defined as a rapid stream of fluid with a small cross-sectional area. When a fluid jet strikes a solid target, the fluid is deflected and flows along the surface of the target. An example of this motion is shown below:

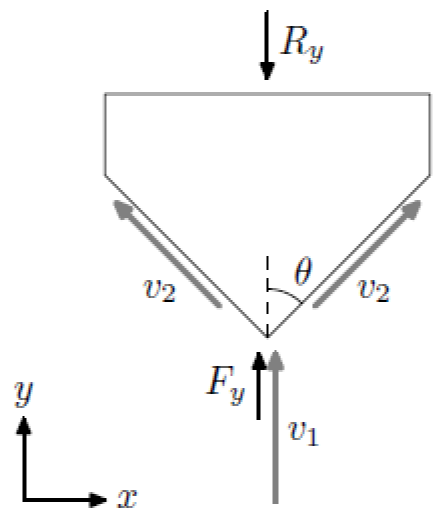



Figure 3.1 Example of a water jet striking a solid surface (from Prof. Huidae Cho's lab manual)

In the figure above, the fluid stream has a velocity $\mathrm{v}_1$; after impacting the solid surface the subsequent streams have a velocity of $\mathrm{v}_2$. The impact force of the fluid on the surface is $F_y$ and the reaction force is $R_y$. For this experiment, we'll consider a jet of water.

The impact force can be determined by taking into account the rate of change of momentum with respect to time. Applying Newton's second law of motion, we can quantify the rate of change of momentum as a function of the fluid's density, flow rate and the change in linear velocity.

$$F_y = \frac{d(m\mathrm{v})}{dt} = \frac{\rho V d\mathrm{v}}{dt} = \rho Q d\mathrm{v} \tag{3.1}$$

Assuming no losses due to friction or shock, the magnitude of the jet velocity does not change after impact, which allows us to say $\mathrm{v}_1 = \mathrm{v}_2 = \mathrm{v}$. However, part of the momentum of the water jet is lost as a result of the impact. Keeping in mind that the reaction force has the same magnitude as the impact force, we can quantify the momentum loss as part of the impact force as:

$$R_y = -F_y = -\rho Q(\mathrm{v}_2 cos\theta - \mathrm{v}_1) = \rho Q \mathrm{v}(1 - cos\theta) \tag{3.2}$$

**Procedure**

This experiment will use an instrument that measures jet impact on a solid surface. Figure 3.2 shows the experiment setup:

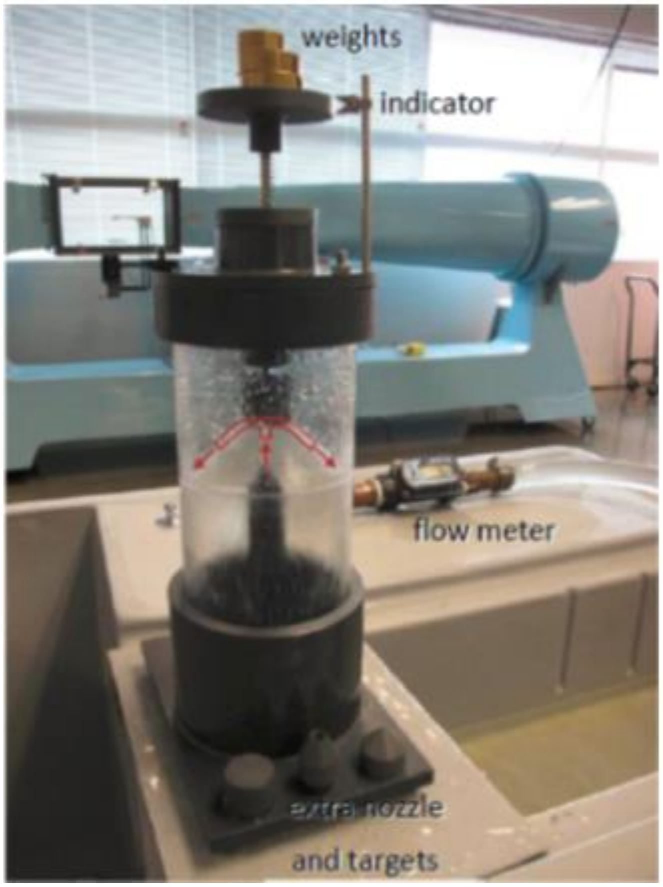


Figure 3.2 Experiment setup (from Prof. Kevin McFall's laboratory manual)

The equipment includes three targets with different angles: convex (45°), flat (90°), and concave (135°), as well as two nozzles with openings of 5mm and 8mm diameter.

1. Level the instrument and make sure that the instrument is firmly placed on the hydraulic bench.

2. Adjust the flow to minimum for startup and turn off the pump once adjusted.

3. Remove the top cover of the instrument. Attach the 5mm nozzle and the convex 45° target on the instrument.

4. Measure the distance (Δy) between the tip if the nozzle and the target. Replace the instrument cover.

5. Position the weight carrier on the weight platform. Add weight until the target is free of the stop and is about midpoint between target and nozzle.

6. Record the weight on the carrier. Align the pointer with the weight platform for reference.

7. Increase flow slowly. Check and adjust the orientation of the jet and target. Increase flow slowly to the maximum readable value.

8. Place additional weights on the platform to return to its initial position. Record the new weight and the flow rate.

9. Reduce the flow rate in steps; adjust the weights on the platform to return to its initial position with each move.

10. Obtain data for 5 different flows.

11. Install the flat 90° target and repeat the procedure.

12. Install the concave 135° target and repeat the procedure.

13. Install the 8mm nozzle and repeat the procedure for all targets.

**Results**

1. Create six three-column tables that show your distance, flow and mass measurements for the 5- and 8-mm nozzles and the 45- 90- and 135° targets, respectively. For this table, show your flow in cms, your mass in kg, and your distance in meters. (Remember to show at least 1 sample computation for every conversion or to provide a table of conversion factors in the Appendix, if applicable).

2. Add two columns to each of your tables showing nozzle diameter (in meters) and nozzle area (in square meters).

3. Compute the nozzle velocity (remember that $\mathrm{Q} = \mathrm{vA}$ and we are assuming no losses from the flow meter to the nozzle). Add a column to each of your tables with the nozzle velocity (in mps)

4. Use the Bernoulli equation to find the actual velocity of the fluid as it hits the nozzle.

$$\frac{\mathrm{P}_1}{\gamma} + \frac{\mathrm{v}_1^2}{2g} + z_1 = \frac{P_2}{\gamma} + \frac{\mathrm{v}_2^2}{2g} + z_2 \tag{3.3}$$

5. Compute the experimental force using the data from the mass of the platforms:

6. Compute the theoretical force using equation 3.2.

7. Add a column to each of your tables with the theoretical force (in N). In total, each table should have 9 columns

8. Plot the experimental force versus the initial fluid force ($\rho Q$v). You should have two plots: one for the 5mm nozzle and one for the 8mm nozzle, each with three different datasets, one for each target angle.

9. Determine the slope of each dataset on both plots (you can either find an average or use linear regression, as long as you explain your procedure in the report)

**Discussion**

1. What does the slope of each dataset (in the plot of measured impact force vs. initial force) represent and how is it related to the experiment?

2. Do the slopes agree with the theoretical values? (Hint: You can determine the theoretical slopes by plotting the theoretical impact force vs. initial force. Be sure to identify and number the graphs if you use this method.)

3. Discuss how the slopes vary according to the target angle from a theoretical perspective (using math) and experimental perspective (using your measurements). Discuss how the slopes vary according to the nozzle size from a theoretical and experimental perspective.

**Analysis**

1. Based on the assumptions of the theory, discuss what would happen in relation to the impact force and experiment procedure, if the jet is not hitting exactly at the center of the target.

2. Discuss the significance of $\Delta y$. Based on your measurements and computations, do you think the velocity correction could have been neglected? How would your final results have been affected if you had assumed no velocity loss in the experiment? Would you still arrive at the same conclusions on your discussion?

3. If you think that the velocity correction can be neglected for this experiment, recommend a minimum height at which the velocity correction is necessary. You may use computations to support your recommendations. OR If you think that the velocity correction cannot be neglected for this experiment, recommend a maximum height at which the velocity correction may be obviated. You may use computations to support your recommendations.

**Conclusion**

1. Formulate and present an empirical relationship between the angle of deflection and impact force.

**Sample Data Sheet**

Experiment 3: Free Jets

5mm Nozzle

| Convex 45° | | | Flat 90° | | | Concave 135° | | |
|---|---|---|---|---|---|---|---|---|
| Δy $(mm)$ | Q $(\times 10^{-4}\ m^3/s)$ | mass $(g)$ | Δy $(mm)$ | Q $(\times 10^{-4}\ m^3/s)$ | mass $(g)$ | Δy $(mm)$ | Q $(\times 10^{-4}\ m^3/s)$ | mass $(g)$ |
| 19.1 | 2.16 | 100 | 22.2 | 1.44 | 100 | 19.1 | 0.801 | 100 |
| | 2.54 | 150 | | 2.09 | 200 | | 1.96 | 200 |
| | 2.97 | 200 | | 2.56 | 300 | | 2.06 | 300 |
| | 3.45 | 250 | | 2.90 | 400 | | 2.35 | 400 |
| | 3.49 | 300 | | 3.15 | 500 | | 2.56 | 500 |

8mm Nozzle

| Convex 45° | | | Flat 90° | | | Concave 135° | | |
|---|---|---|---|---|---|---|---|---|
| Δy $(mm)$ | Q $(\times 10^{-4}\ m^3/s)$ | mass $(g)$ | Δy $(mm)$ | Q $(\times 10^{-4}\ m^3/s)$ | mass $(g)$ | Δy $(mm)$ | Q $(\times 10^{-4}\ m^3/s)$ | mass $(g)$ |
| 28.6 | 2.54 | 50 | 22.2 | 2.47 | 100 | 15.9 | 1.78 | 100 |
| | 3.23 | 60 | | 3.32 | 200 | | 2.44 | 200 |
| | 4.16 | 100 | | 4.06 | 300 | | 3.08 | 300 |
| | 5.03 | 150 | | 4.53 | 400 | | 3.51 | 400 |
| | 5.59 | 170 | | 5.20 | 500 | | 4.03 | 500 |

## Pelton Wheel Turbine

**Objective**

The objective of this experiment is to determine the maximum power and maximum efficiency of a Pelton wheel turbine.

**Theory**

The Pelton wheel is a type of turbine used to capture kinetic energy from fluids and convert it into mechanical energy. This mechanical energy rotates a shaft which converts it into electrical energy. The turbine is designed so that the inlet and outlet velocities of the fluid can be determined by the angle of its blades. A schematic of flow through a Pelton wheel blade is shown below:

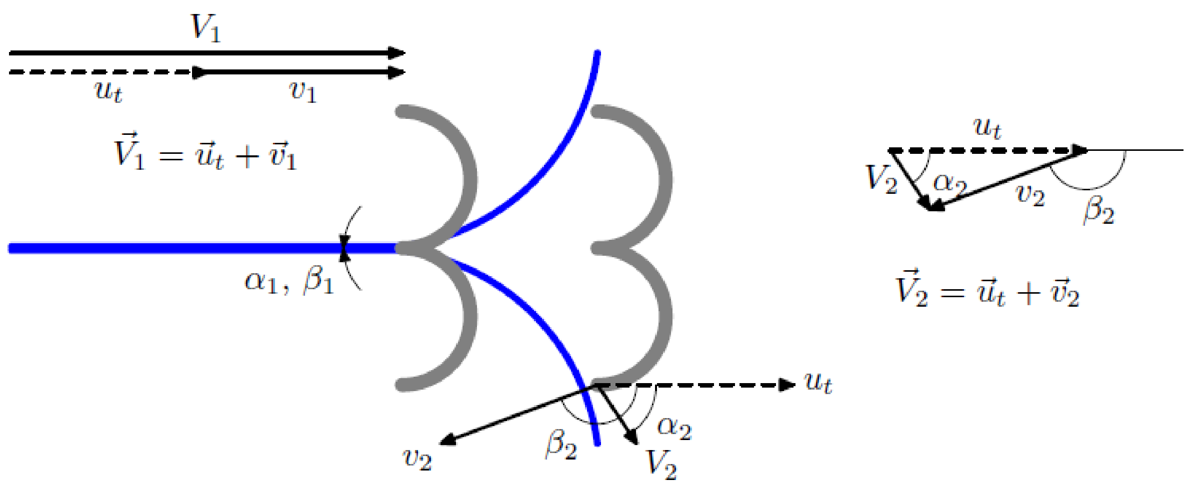



Figure 4.1 Pelton wheel turbine schematic (from Prof. Huidae Cho's Lab Manual)

Figure 4.1 shows a jet striking a Pelton wheel blade and its associated velocity vectors. $\mathrm{v}_1$ and $u_1$ are the absolute and relative velocities of the inlet flow, respectively. $\mathrm{v}_2$ and $u_2$ correspond to the outlet flow. $u_t$ is the tangential velocity of the blade. $\alpha_1$ and $\alpha_2$ are the inlet and outlet angles of the velocity of the flow, respectively. $\beta_1$ and $\beta_2$ are the angles of the blades at the inlet and outlet, respectively.

The main concept behind the Pelton wheel turbine is that a change in the direction between the inflow and outflow momenta of a jet can maximize the impact force on a target. Applying this concept to a turbine, where the targets are the blades on the turbine, we can derive equations from our jet flow equations and translate them into equations for turbine speed.

For a fixed jet velocity and fixed flowrate, we will have a constant angular speed of the turbine. Consequently, the tangential speed of the turbine blades will also be constant. We define tangential velocity as $u_t = \omega r$ where $r$ is the radius of the Pelton wheel. To determine the efficiency of the turbine at different heads, the input and output powers of the turbine have to be known. The power developed by the turbine is a product of the force generated by the fluid. This force can be determined from the calculation of the momentum change in the fluid while in contact with the blade. Therefore, we can rewrite the jet equation for impact force as:

$$-R = \sum F_x = \rho Q(\mathrm{v}_2 \cos(\alpha_2) - \mathrm{v}_1) \tag{4.1}$$

Using geometric relationships, we can determine the velocity components and obtain:

$$u_t = \mathrm{v}_2 \cos(\alpha_2) - u_2 \cos(\beta_2) \tag{4.2}$$

Assuming no friction losses ($u_1 = u_2$), and combining the previous equations, we obtain:

$$-R = \rho Q(-u_1 + u_2 \cos(\beta_2)) = \rho Q u(\cos(\beta_2) - 1) \tag{4.3}$$

The force generated by the fluid then becomes:

$$R = \rho Q u(1 - \cos(\beta_2)) = \rho Q(\mathrm{v}_1 - u_t)(1 - \cos(\beta_2)) \tag{4.4}$$

The input power is computed as the product of the impact force of the fluid and the tangential velocity of the blade:

$$P_{in} = R u_t = \rho Q u_t(\mathrm{v}_1 - u_t)(1 - \cos(\beta_2)) \tag{4.5}$$

where Pin is the input power to the Pelton wheel turbine. There is a theoretical speed that would produce the maximum power, identified as $u_P$. This theoretical speed can be found in relation to $\mathrm{v}_1$ by differentiating the input power equation with respect to $u_t$ and setting it to zero. The output power is computed as the product of the torque of the shaft and the angular velocity:

$$P_{out} = T_{out}\omega = \frac{F_{out} d}{2}\omega \tag{4.6}$$

where $P_{out}$ is the output power, $T_{out}$ is the torque of the shaft, $F_{out}$ is the output force, and $d$ is the diameter of the drum. The output torque and force can be determined experimentally by using spring scales.

In this experiment, a small model of the Pelton wheel turbine with attached spring scales will be used. The output force is determined by:

$$F_{out} = F + \Delta F_1 - (F - \Delta F_2) = \Delta F_1 + \Delta F_2 \tag{4.7}$$

where $F$ is the initial force before the jet flow impact, and $\Delta F_1$ & $\Delta F_2$ are differences in force for the two spring scales, respectively. After the input and output powers are known, the efficiency of the turbine is obtained by taking the ratio of the output power over the input power:

$$\eta = \frac{P_{out}}{P_{in}} \tag{4.8}$$

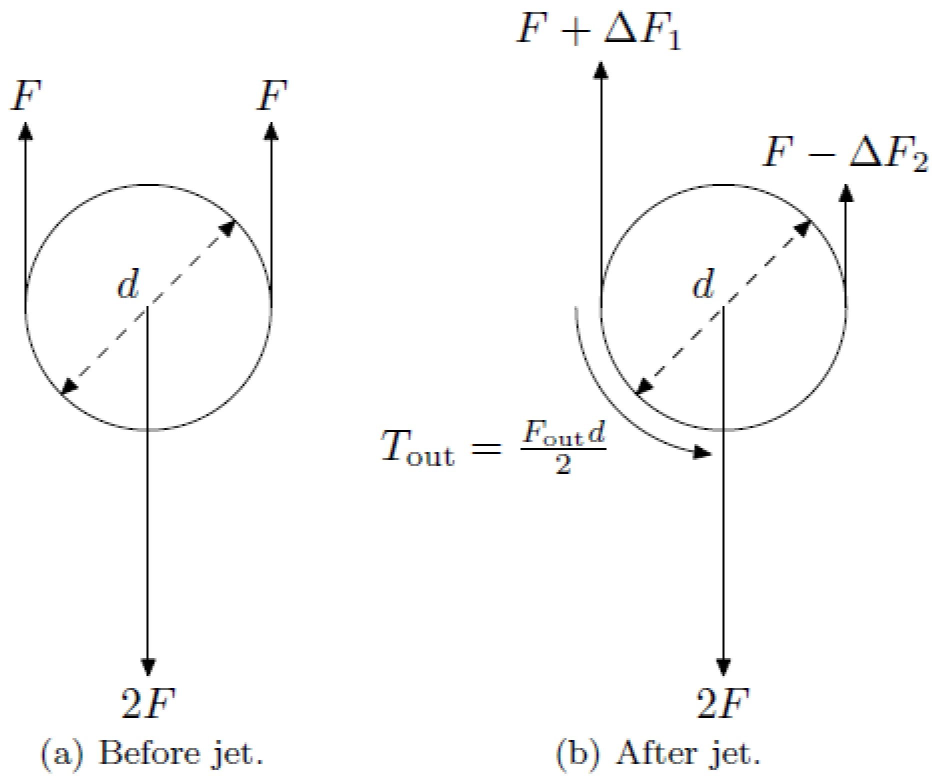



Figure 4.2 Output force diagrams from the Pelton wheel model (from Prof. Huidae Cho's lab manual)

**Procedure**

1. Attach the spring scales and polyester fabric to the drum of the Pelton wheel turbine.

2. Familiarize yourself with the function of the digital tachometer. Attach the top of the tachometer to the center of the drum to take readings of angular speed in RPM.

3. Turn on the pump on the hydraulic bench. Fully open the valve on the hydraulic bench and adjust the nozzle on the Pelton wheel turbine such that the pressure reading on the pressure gage is maxed out. If the jet is not hitting the bucket directly in the center, adjust the nozzle again to obtain a pressure reading of about 1.5 bar.

4. Turn off the pump.

5. Tighten the spring scales and fabric by adjusting the knob on top of the metal frame. Record the initial reading of both scales.

6. Turn on the pump and allow the wheel to spin for a few seconds. Record the reading of the scales again. After one data point is obtained, turn off the pump

7. Repeat steps 5-6 for five different tension settings.

8. Change the pressure setting and repeat steps 5-7 again. Collect data for a total of 4 different pressures.

**Results**

For this experiment, $\beta_2 = 165°$.

1. Compute the input and output powers and efficiencies for all your data.

2. Derive the v1/ut ratio that will produce the maximum power.

3. Plot Pin vs. v1/ut (fit quadratic trend lines for your plots) for all nozzle settings.

4. Plot η vs. RPM and Torque vs. RPM. Fit quadratic trend lines for your plots.

**Discussion**

1. Compare the peaks to the v1/ut ratio you derived in step 2 of the procedure.

2. Compare the RPM for the maximum torque and the RPM for the maximum efficiency. Does the maximum torque and maximum efficiency RPM line up? Discuss your readings.

3. Discuss the difference between the point of the maximum efficiency and the point of the maximum output power from your data. Do they correlate?

**Analysis**

1. Can the results from this experiment be applied to a Pelton Wheel Turbine that is larger in size but maintains the same proportions and deflection angles? Why or why not?

**Conclusion**

1. Determine the maximum power and maximum efficiency of this Pelton wheel turbine.

2. Explain the application of measurements taken from this Pelton Wheel turbine into larger models.

**Sample Data Sheet**

Experiment 4: Pelton Wheel Turbine

Specifications: Drum diameter, d: 2.5 in, Pipe diameter: 1.5 in, Runner radius, r: 2.5 in.

| Pressure (bar) | Q (gpm) | ω (RPM) | Initial R1 | Initial R2 | Final R1 | Final R2 |
|---|---|---|---|---|---|---|
| 0.7 | 12.27 | 1030 | 6 | 4 | 10 | 0 |
| 0.7 | 12.38 | 750.4 | 7.5 | 6 | 13 | 0.5 |
| 0.7 | 12.27 | 552.2 | 10 | 8.5 | 16.5 | 2 |
| 0.7 | 12.38 | 490.1 | 12 | 10.25 | 19 | 4 |
| 0.7 | 12.4 | 419 | 13.5 | 12 | 20 | 7.5 |
| 1 | 11.34 | 1500 | 6 | 4 | 10 | 0 |
| 1 | 11.02 | 981.3 | 7.5 | 6 | 13.5 | 0.25 |
| 1 | 11.32 | 596 | 10 | 8.5 | 16.5 | 2 |
| 1 | 10.94 | 590.9 | 12 | 10.25 | 19 | 4 |
| 1 | 11.34 | 458 | 13.5 | 12 | 21 | 5 |
| 1.5 | 9.01 | 2118 | 12 | 10 | 19 | 3 |
| 1.5 | 9.06 | 960 | 8 | 6 | 4 | 1 |
| 1.5 | 9.05 | 650 | 10 | 8 | 16 | 2 |
| 1.5 | 9.03 | 470 | 13 | 11 | 20 | 4 |
| 1.5 | 9.08 | 370 | 14 | 12 | 21 | 5 |
| 2 | 6.03 | 770 | 6 | 4 | 10 | 0 |
| 2 | 5.28 | 331.6 | 7.5 | 6 | 12 | 2 |
| 2 | 5.72 | 201.3 | 10 | 8.5 | 15.5 | 3.5 |
| 2 | 5.22 | 196.7 | 12 | 10.25 | 17 | 6 |
| 2 | 5.35 | 112.8 | 13.5 | 12 | 18.5 | 7 |

# Closed Conduit Flow

**Objective**

The objectives of this experiment are (1) to understand how pressure drops relate to losses in energy, (2) to show how to measure major and minor losses experimentally, and (3) to determine friction factors and loss coefficients experimentally.

**Theory**

We can analyze losses in a closed conduit system using the energy equation, which can relate the energy between two cross sections. The main terms in the energy equation consist of the pressure head (pressure energy), velocity head (kinetic energy) and elevation head (potential energy). These terms are shown in the order mentioned as below:

$$E = \frac{P}{\gamma} + \frac{\mathrm{v}^2}{2g} + z \tag{5.1}$$

where $E$ is the total energy, $P$ is the pressure, $\gamma$ is the specific weight of fluid, $\mathrm{v}$ is the fluid velocity, $g$ is the gravitational acceleration, and $z$ is the elevation. As fluid travels from one cross section to another downstream within a pipe, friction and component losses between the two cross sections will reduce the amount of energy at the downstream cross section. Therefore, the total energy at downstream will be the same as the total energy at upstream minus all losses in-between. This relationship can be expressed as follows:

$$E_1 - h_L = E_2 \tag{5.2}$$

where $E_1$ and $E_2$ are the total energy at upstream and downstream cross sections, respectively, and $h_L$ is the total loss or head loss between the two cross sections. The head loss consists of the friction loss $h_f$ and the component loss $h_m$ as follows:

$$h_L = h_f + h_m \tag{5.3}$$

In a system where the diameter of a pipe does not change in size and the elevations of upstream and downstream cross sections are the same, the velocity and elevation heads remain constant. Whenever there are any losses, the pressure head will experience a decrease in magnitude. Therefore, in such a scenario, the loss terms can be represented by the following simple relationship:

$$\frac{\Delta P}{\gamma} = h_L = h_f + h_m \tag{5.4}$$

In many textbooks, the friction loss is also known as major loss while the component loss is also called minor loss. The magnitude of these losses can be measured and is dependent upon the passing fluid velocity. In addition to the flow rate, the friction loss depends on the roughness of the pipe carrying the flow, the diameter of the pipe, the distance between cross sections, and the state of flow (laminar or turbulent). The combined effects of the state of flow and material roughness can be simplified to a coefficient called friction factor $f$. If the loss within a system is purely due to friction, the friction loss can be represented as follows:

$$h_f = \frac{\mathit{\Delta P}}{\gamma} = f\,\frac{L}{D}\frac{\mathrm{v}^2}{2g} \tag{5.5}$$

where $L$ is the length of the pipe between two cross sections and $D$ is the diameter of the pipe. Similarly, the minor loss can be represented as follows:

$$h_m = (\sum k_i)\frac{\mathrm{v}^2}{2g} \tag{5.6}$$

where $k$ is the minor loss coefficient. In a system where both major and minor losses are present, the head loss equation is thus a combination of major and minor losses as follows:

$$h_L = h_f + h_m = f\,\frac{L}{D}\frac{\mathrm{v}^2}{2g} + \ (\sum k_i)\frac{\mathrm{v}^2}{2g} \ = \left[f\,\frac{L}{D} + \ (\sum k_i)\right]\frac{\mathrm{v}^2}{2g} \tag{5.7}$$

By substituting $\mathrm{v} = \frac{\mathrm{Q}}{\pi\mathrm{D}^2/4}$ where Q is the flow rate, and relating the total loss back to the pressure drop, we obtain the following:

$$\frac{\mathit{\Delta P}}{\gamma} = h_l = h_f + h_m = \left[f\,\frac{L}{D} + \ (\sum k_i)\right]\frac{8Q^2}{\pi^2 g D^4} \tag{5.8}$$

You may use the Moody diagram or an empirical equation such as the Swamee-Jain equation to estimate the friction factor:

$$\frac{1}{\sqrt{f}} = -2\log_{10}\left[\frac{\varepsilon/D}{3.7} + \frac{5.74}{\mathrm{Re}^{0.9}}\right] \tag{5.9}$$

where ε is the roughness height, D is the pipe diameter, and Re is the Reynolds number, which is defined as follows:

$$\mathrm{Re} = \frac{\mathrm{v}D}{\nu} \tag{5.10}$$

where v is the mean velocity of the fluid, $D$ is the diameter of the pipe, and $\nu$ is the kinematic viscosity of the fluid.

**Procedure**

1. Log on to the computer and start AFTC.
2. Click on the start button and accept the file location to save data.

3. Activate the actuator (AB-1 button) on screen to start the pump.

4. Ensure that only the appropriate valves are opened and connect the manometer tubes on the network.

5. For experiment 1 (friction loss, rough pipe with no minor losses), measure the length of the pipe across the two manometer tubes and record the diameter of the pipe. For each flow rate, record the difference in pressure head in mm. Record data for at least 8 readings at different flow rates.

6. For experiment 2 (minor loss, smooth pipe with a ball valve half open), measure the length and diameter of the pipe. Open the ball valve only half and record the difference in pressure head in mm for 4 different flow rates.

7. For experiment 3 (split flow, smooth pipe with split flow), identify and count the minor loss components (elbows, tees, valves, etc.). Measure the lengths and diameters of the top and bottom pipes. Open the ball valve only half and records the difference in pressure head in mm for 4 different flow rates.

**Results**

Use $\varepsilon = 0.175mm$ for your calculations.

Friction loss (rough pipe with no minor losses)

1. Create a table for the friction loss data that contains the flow rate (in cubic meters per second) and the pressure head (in meters).

2. For every flow rate in the table, compute the corresponding velocity.

3. Use the Eq. 5.5 to compute the friction factors for all your data.

4. Calculate the Reynolds number for each measurement.

5. Plot the friction factor vs. the Reynolds number.

Minor loss (smooth pipe with a ball valve half open)

By rearranging Eq. 5.8, we can obtain the following equation:

$$\frac{\Delta P}{\gamma} - h_f = \frac{\Delta P}{\gamma} - f\frac{L}{D}\frac{8Q^2}{\pi^2 g D^4} = \left(\sum k_i\right)\frac{8Q^2}{\pi^2 g D^4} = CQ^2 \qquad (5.11)$$

where $C = \left(\sum k_i\right)\frac{8}{\pi^2 g D^4}$

1. Create a table for the minor loss data and include the flow rate (in$m^3/s$), and the pressure head (in meters).
2. For every flow rate in the table, compute the velocity.
3. Calculate the Reynolds number for each measurement.
4. Use Eq. 5.9 to estimate the friction factors for all your data.
5. Calculate the major losses using Eq. 5.5
6. Plot a graph of $\frac{\Delta P}{\gamma} - h_f$ vs. Q
7. Fit a quadratic curve with the form $y = Cx^2$ through the plot. You can use the LINEST function in Excel. Show your curve and parameters on the figure.
8. Use the value of $C$ from your fit to determine the minor loss coefficient $k$ value of the half open ball valve.

Split flow (smooth pipe with split flow)

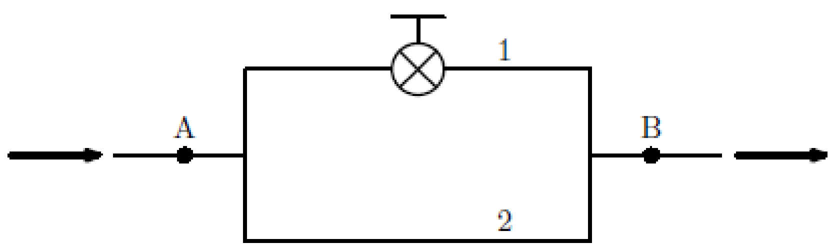



Figure 5.1 Parallel pipe section with a ball valve on the upper section of the pipe and pressure taps at A and B. (From Prof. Huidae Cho's lab manual)

The total flow rate $Q$ is the sum of the flow rate for the top loop $Q_1$ and the flow rate for the bottom loop $Q_2$:

$$Q = Q_1 + Q_2 \tag{5.12}$$

Also, the head difference between A and B should be the same across the top loop 1 and the bottom loop 2:

$$h_L = h_{L_1} = h_{L_2} \tag{5.13}$$

1. Determine the flow rates for the two parallel pipes using the minor loss coefficients for the half open ball valve, elbows, and tees from the Engineering ToolBox or the Fluid Mechanics textbook. Since the Reynolds number in Eq. 5.9 is a function of the flow velocity or flow rate, in total, there are two unknown variables $Q_1$ and $Q_2$, and two equations (5.12 and 5.13). Therefore, you can directly solve these equations for $Q_1$ and $Q_2$. However, since the flow rate in the Reynolds number is in a log form in 5.9, you may need to use

the trial and error method as follows:

a. Obtain $k$ values from literature.

b. Guess $Q_1$ and calculate $Q_2$ from your guess

c. Calculate $\mathrm{v}_1$ and $\mathrm{v}_2$

d. Estimate $f_1$ and $f_2$ using equation 5.9

e. Calculate $h_{L_1}$ and $h_{L_2}$

f. Repeat these steps, guessing different initial $Q_1$ until $\Delta h_L = |h_{L_1} - h_{L_2}|$ is close to zero

**Discussion**

<u>Friction loss (rough pipe with no minor losses)</u>

1. Discuss the trend of the friction factor vs. the Reynolds number.
2. Does your friction factor vs. Reynolds number curve agree with the Swamee-Jain equation? To present your comparison, plot a curve of the friction factor vs. Reynolds number using the Swamee-Jain equation and compare it to the figure from the Results section. Be sure to label and describe the curve.

<u>Minor loss (smooth pipe with a ball valve half open)</u>

1. Compare your computed value of k to the published value of the half open ball valve.

<u>Split flow (smooth pipe with split flow)</u>

1. After obtaining $Q_1$ and $Q_2$, compare theoretical head losses across both loops to the pressure head difference you obtained experimentally.

**Analysis**

<u>Friction loss (rough pipe with no minor losses)</u>

1. To compute the friction factors, you are assuming that the only losses in the system are due to pressure head. Explain why it is safe to assume that there will be no kinematic or potential losses in the system.

Minor loss (smooth pipe with a ball valve half open)

1. What assumptions need to be made in order to run the minor loss experiment? Do you think these assumptions can compromise the accuracy of the final results?

Split flow (smooth pipe with split flow)

1. In order to obtain the head losses for this part, a trial-and-error method has to be implemented as explained in the Results section. To do this, the difference between $h_{L_1}$ and $h_{L_2}$ has to be minimized. Suggest an appropriate tolerance value for this difference that will not affect the final results significantly. For this part, you can either mathematically find a relationship between the $\Delta h_L$ error and the resulting error in flow; or alternatively you can write a code in your preferred programming language to carry out the trial-and-error method explained in the Results section, and vary the tolerance of $\Delta h_L$ to find the resulting errors in flow.

**Conclusion**

1. Is the Swamee-Jain equation appropriate to determine the friction factor of a pipe? How does it compare to using the Moody diagram or finding the friction factor from measurements?

2. Can the minor loss coefficient calculated in the second part be used for any type of valve? Why or why not?

3. Recommend the best methodology for calculating flow distribution in pipe networks.

**Sample Data Sheet**

Experiment 5: Closed Conduit Flow

Friction Loss experiment

Length: 1.00 m

Diameter: 17 mm

| Flow rate (gpm) | Pressure head (mm) |
|---|---|
| 15.2 | 470 |
| 15.0 | 457 |
| 14.6 | 432 |
| 14.3 | 417 |
| 13.7 | 389 |
| 12.7 | 343 |
| 10.8 | 269 |
| 9.90 | 239 |

Minor Loss experiment

Length: 0.146 m

Diameter: 17 mm

| Flow rate (gpm) | Pressure head (mm) |
|---|---|
| 14.5 | 546 |
| 13.8 | 503 |
| 13.0 | 457 |
| 10.4 | 320 |

Split Flow experiment

Top branch

Length: 0.559 m

Diameter: 17 mm

Components:

tees (2)

elbows (2)

ball valve (1)

| Flow rate (gpm) | Pressure head (mm) |
|---|---|
| 15.0 | 203 |
| 14.2 | 188 |
| 13.4 | 168 |
| 10.8 | 119 |

Bottom branch

Length: 0.559 m

Diameter: 17 mm

Components:

tees (2)

elbows (2)

## Pump Performance

**Objective**

The objective of this experiment is to find the most efficient pump setting for a given pump and system.

**Theory**

In order to pump water from one location to another, a pump needs to overcome the energy difference between the two points. If we identify the inlet and outlet locations of a control volume as 1 and 2, respectively, then the energy at the inlet can be expressed as:

$$E_1 = \frac{P_1}{\gamma} + \frac{\mathrm{v}_1^2}{2g} + z_1 \tag{6.1}$$

Similarly, the energy at the outlet can be expressed as:

$$E_2 = \frac{P_2}{\gamma} + \frac{\mathrm{v}_2^2}{2g} + z_2 \tag{6.2}$$

The energy equation for steady uniform flow in the entire control volume can be rearranged to solve for pump head, which yields:

$$h_P = E_2 - E_1 + h_L = \frac{P_2 - P_1}{\gamma} + \frac{\mathrm{v}_2^2 - \mathrm{v}_1^1}{2g} + z_2 - z_1 + h_L \tag{6.3}$$

Losses in a closed conduit system can be quantified using the energy equation. Normally, the losses are separated into friction (major) and component (minor) losses. Friction losses can be computed using the Darcy-Weisbach equation:

$$h_f = f\frac{L}{D}\frac{\mathrm{v}^2}{2g} \tag{6.4}$$

where $f$ is the friction factor, $L$ and $D$ are the length and diameter of the pipe, respectively, and v is the velocity. The friction factor can be either read from the Moody diagram or calculated using empirical equations. One such empirical equation is the Swamee-Jain equation:

$$f = 0.25\left[\log_{10}\left(\frac{\varepsilon}{3.7D} + \frac{5.74}{\mathrm{Re}^{0.9}}\right)\right]^{-2} \tag{6.5}$$

where $\varepsilon$ is the material roughness height and Re is the Reynolds number. The Reynolds number is defined as:

$$\mathrm{Re} = \frac{\mathrm{v}D}{\nu} \tag{6.6}$$

where $\nu$ is the kinematic viscosity. For this experiment, we will ignore the minor losses and only consider friction loss, which reduces the pump head equation to:

$$h_P = \frac{\Delta P}{\gamma} + \frac{\mathrm{v}_2^2 - \mathrm{v}_1^1}{2g} + \Delta z + f\frac{8LQ^2}{\pi^2 g D^5} \quad (6.7)$$

A system curve is a curve used to display the relationship between the system's required pump head and the flow rate, $Q$.

Similar to the system curve, a pump curve characterizes the relationship between the pump's supplied pump head and the flow rate. Typically, the pump curve is provided by the manufacturer. In this experiment, the pump curve will be obtained experimentally. A pump may have different efficiencies for different flows. The point along a pump curve that coincides with the maximum efficiency is often called the Best Efficiency Point (BEP). An operating point closest to the BEP is regarded as a good operating point due to the fact that the pump is delivering flow closest to its maximum efficiency.

The brake horsepower, $P_{brake}$ before any loss in power can be written as:

$$P_{brake} = T\omega \quad (6.8)$$

where $T$ and $\omega$ are the torque and angular speed of the shaft, respectively. Due to friction and minor losses in the pump, the actual power used to deliver water will be less than the brake horsepower. The water horsepower $P_{water}$ can be defined as:

$$P_{water} = F\mathrm{v} = \Delta P A\mathrm{v} = \gamma h_P A\mathrm{v} = \gamma h_P Q \quad (6.9)$$

where $F$ and v are the force and velocity of water, respectively, $\Delta P$ is the change in pressure across the pump, $A$ and $Q$ are the flow area and rate, respectively, and $h_P$ is the machine head or pump head. The efficiency of the pump can be obtained by dividing $P_{water}$ over $P_{brake}$.

When the pump curve and system curve are plotted together on the same graph (as shown in Figure 6.1), the point of intersection of the two curves will be the operating point. The operating point will change if a different pump is used. Therefore, choosing the pump that produces an operating point close to its BEP will ensure that the pump will operate at an efficient head and flow rate.

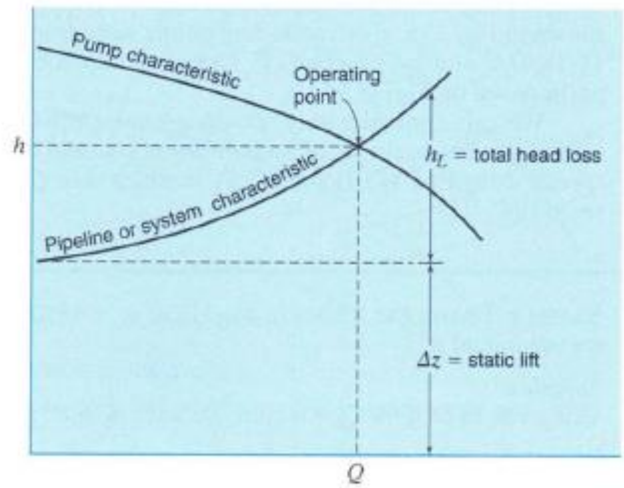



Figure 6.1 System and pump curves, and operating point. From Finnemore and Franzini, Fluid Mechanics with Engineering Applications 10th Ed.

**Procedure**

1. Log into the PC and start PB2C.
2. Turn on the switch located on the PB2C box next to the PC.
3. Open the valve for the centrifugal pump only. Shut the valve for the gear pump completely.
4. Select the centrifugal pump as your operational pump.
5. Select the location where you would like to save data.
6. Set the actuator to 3000 RPM. Allow 60 seconds for the flow to stabilize.
7. Select the data table. Select Average and Acquire.
8. Partially close the valve to reduce the flow by approximately 10 L/min. Obtain data for 5 different flow rates.
9. Repeat steps 5-7 for shaft speeds of 2750, 2500, and 2000 RPM.
10. Stop and quit the program.
11. Record the data table.

**Results**

For the purpose of this lab, a hypothetical scenario is created so that a system curve can be obtained. Assume that you have a system that delivers water from a lower reservoir (1) at elevation $z_1 = 100\ m$, to another reservoir (2) at $z_2 = 105m$ with a 20° incline using a 1.5 $cm$ diameter pipe made of commercial steel (surface roughness $\varepsilon = 0.046\ mm$). The two reservoirs are stagnant ($v_1 = v_2 = 0$) and all minor losses are negligible.

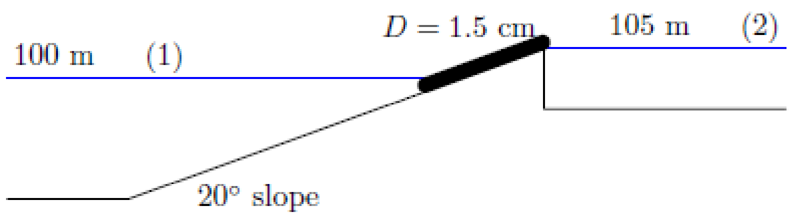



Figure 6.2 Hypothetical system. From Prof. HuidaeCho's lab manual.

1. Plot the system curve ($h_P$ vs. $Q$), fit a curve into the plot.

2. Plot the pump curve ($h_P$ vs. $Q$) and efficiency curve (efficiency vs. $Q$) for each pump speed. Fit a curve into each pump curve and efficiency curve. Quantitatively identify the BEP.

3. Join the pump, efficiency, and system curves. You can plot them together in Excel. You should have four figures (one for each RPM= 3000, 2750, 2500, 2000) each with three curves: system curve, pump curve, and efficiency curve.

4. Identify the operating point for each pump speed.

5. Identify and report the efficiency associated with the operating point.

6. Discuss your findings

**Conclusion**

1. Recommend the best RPM setting for the pump.

**Sample Data Sheet**

## Expt #: 6 - Data for Pump Performance

| SL No. | RPM | $P_{adm}$ (bar) | $P_{dis}$ (bar) | Level (mm) | Flow (L/min) | Torque (N-m) | Machine Head (Ht - m) | Water horsepower (Pout - Nm/s) | Brake hosrepower (Pin - Nm/s) | Efficiency (%) |
|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 3000 | | | 26.373 | 26.062 | 0.464 | 10.374 | 44.203 | 145.78 | 30.322 |
| 2 | 3000 | | | 24.578 | 23.446 | 0.409 | 11.325 | 43.413 | 128.66 | 33.742 |
| 3 | 3000 | | | 19.066 | 16.02 | 0.335 | 12.221 | 32.008 | 105.12 | 30.449 |
| 4 | 3000 | | | 16.607 | 13.023 | 0.334 | 12.245 | 26.072 | 104.93 | 24.847 |
| 5 | 3000 | | | 11.853 | 7.85 | 0.293 | 12.607 | 16.185 | 92.283 | 17.538 |
| 6 | 2750 | | | 25.373 | 24.062 | 0.364 | 9.374 | 42.203 | 142.78 | 29.558 |
| 7 | 2750 | | | 22.578 | 21.446 | 0.309 | 10.325 | 41.413 | 125.66 | 32.956 |
| 8 | 2750 | | | 16.066 | 14.02 | 0.215 | 11.221 | 30.008 | 102.12 | 29.385 |
| 9 | 2750 | | | 12.607 | 11.023 | 0.204 | 11.245 | 24.072 | 100.93 | 23.850 |
| 10 | 2750 | | | 6.853 | 5.85 | 0.153 | 11.607 | 14.185 | 86.283 | 16.440 |
| 11 | 2500 | | | 21.867 | 19.676 | 0.293 | 7.317 | 23.541 | 76.708 | 30.689 |
| 12 | 2500 | | | 18.135 | 14.86 | 0.225 | 8.186 | 19.888 | 59.096 | 33.654 |
| 13 | 2500 | | | 14.282 | 10.386 | 0.176 | 8.662 | 14.709 | 46.207 | 31.833 |
| 14 | 2500 | | | 10.252 | 6.316 | 0.165 | 8.774 | 9.061 | 43.151 | 20.998 |
| 15 | 2500 | | | 5.347 | 2.379 | 0.148 | 8.894 | 3.461 | 38.967 | 8.882 |
| 16 | 2000 | | | 17.972 | 14.66 | 0.169 | 4.673 | 11.201 | 35.424 | 31.620 |
| 17 | 2000 | | | 15.186 | 11.388 | 0.102 | 5.333 | 9.929 | 21.263 | 46.696 |
| 18 | 2000 | | | 10.497 | 6.544 | 0.111 | 5.322 | 5.698 | 33.347 | 17.087 |
| 19 | 2000 | | | 7.999 | 4.353 | 0.091 | 5.231 | 3.933 | 18.969 | 20.734 |
| 20 | 2000 | | | 2.739 | 0.8723 | 0.017 | 5.912 | 0.844 | 15.033 | 5.614 |

# Open Channel Flow

**Objectives**

The objectives of this experiment are (1) to compare different methods of calculating flowrate and recommend the best one, (2) to develop an empirical relationship between flow and depth for a weir.

**Theory**

Open channel flow occurs when a fluid has a free surface open to the atmosphere. In a case like this, the pressure acting on the fluid is the atmospheric pressure and therefore, the energy equation can be written as:

$$E = \frac{\mathrm{v}^2}{2g} + y \tag{7.1}$$

where we now choose $y$ as the depth coordinate. The flow rate in an open channel can be estimated using Manning's equation:

$$Q = \frac{k}{n} A\, R^{2/3} S^{1/2} \tag{7.2}$$

where $Q$ is the flow rate, $k$ is a unit conversion constant (1 for SI units, 1.486 for English units), $n$ is the Manning's roughness coefficient, $A$ is the flow area, $R$ is the hydraulic radius ($R = A/P$), $P$ is the wet perimeter, and $S$ is the friction slope, which is equivalent to the channel slope at normal depth.

The flow rate can also be estimated by applying conservation of mass through a control surface with a constant or average velocity:

$$Q = \mathrm{v}A \tag{7.3}$$

Since this equation can be found by averaging the continuity equation over a control surface, it is sometimes referred to as the continuity equation.

A weir can be defined as a barrier in a hydraulic system that forces the fluid to flow over it into a lower level. The flow through a weir is estimated differently than the flow through a channel. Empirical equations can be obtained with the following form:

$$Q = KH^{\alpha} \tag{7.4}$$

where $K$ and $\alpha$ are constants obtained empirically, and $H$ is the head above the weir. For this experiment, a V-notch weir will be used, and its equation is:

$$Q = \frac{8}{15} C_d \sqrt{2g}\, tan\left(\frac{\theta}{2}\right) H^{5/2} \tag{7.5}$$

where $C_d$ is the discharge coefficient and $\theta$ is the angle of the weir. For a $60°$ weir, the discharge coefficient is $0.5767$.

**Procedure**

To measure flow rate:

1. Measure the heights of the upstream and downstream ends of the channel; also, measure the distance between the two sections. Calculate the slope of the channel.
2. Measure the width of the channel.
3. Turn on the pump and start running water at a flow rate that will produce the highest uniform depth along a stretch of the channel (approx. $30\ gpm$). Wait for 1 minute for the flow to achieve a stready state.
4. Record the depth at the uniform section as well as the average flow rate shown on the flow meter.
5. Select a segment with a uniform depth and mark the upstream and downstream ends.
6. Measure the distance between the two sections and the average depth.
7. Drop a floating device on the surface of the water and use a stopwatch to obtain the travel time between the two sections.
8. Repeat the experiment 4 more times for lower flow rates.

To measure weir flow:

1. Insert the $60°$ weir into the slot in the channel.
2. Start the pump with a low flow rate (approx. $10\ gpm$). Wait for a few minutes for the flow to achieve a stead state.
3. Measure the head above the weir and record the flow rate from the flow meter.
4. Repeat the experiment 4 more times for higher flow rates.

**Results**

For Manning's equation, use $n = 0.010$ for glass

1. For each measured height, compute the flow rate using Manning's equation
2. Compute the flow rate using the continuity equation.
3. Compare the flow rates obtained from the flow meter and Manning's and continuity equations.

4. Compute the percent errors.

Weir flow

1. Plot a graph of Q vs. H and fit a curve using a power trend line. Use the flow rate from the flow meter (Q).

2. Determine the coefficients $K$ and $\alpha$ from equation 7.4 using your measured data.

**Discussion**

Flow rate

1. Discuss why the flow rate from the continuity equation is greater than the flow rate from Manning's equation and the flow meter.

Weir flow

1. Compare the general weir equation (7.4) with the V-notch weir equation (7.5). Do the constant and exponents compare well?

2. Estimate the flow rate using equation 7.5 and compare with your measured flow rates.

**Analysis**

Flow rate

1. Discuss why Manning's equation would be the better option to use in estimating the flow rate in this lab.

Weir flow

1. Compare the flow rates with the flow rate from the flow meter. Assuming that the V-notch weir equation is correct, compute the percent error of the flow rate from the flow meter.

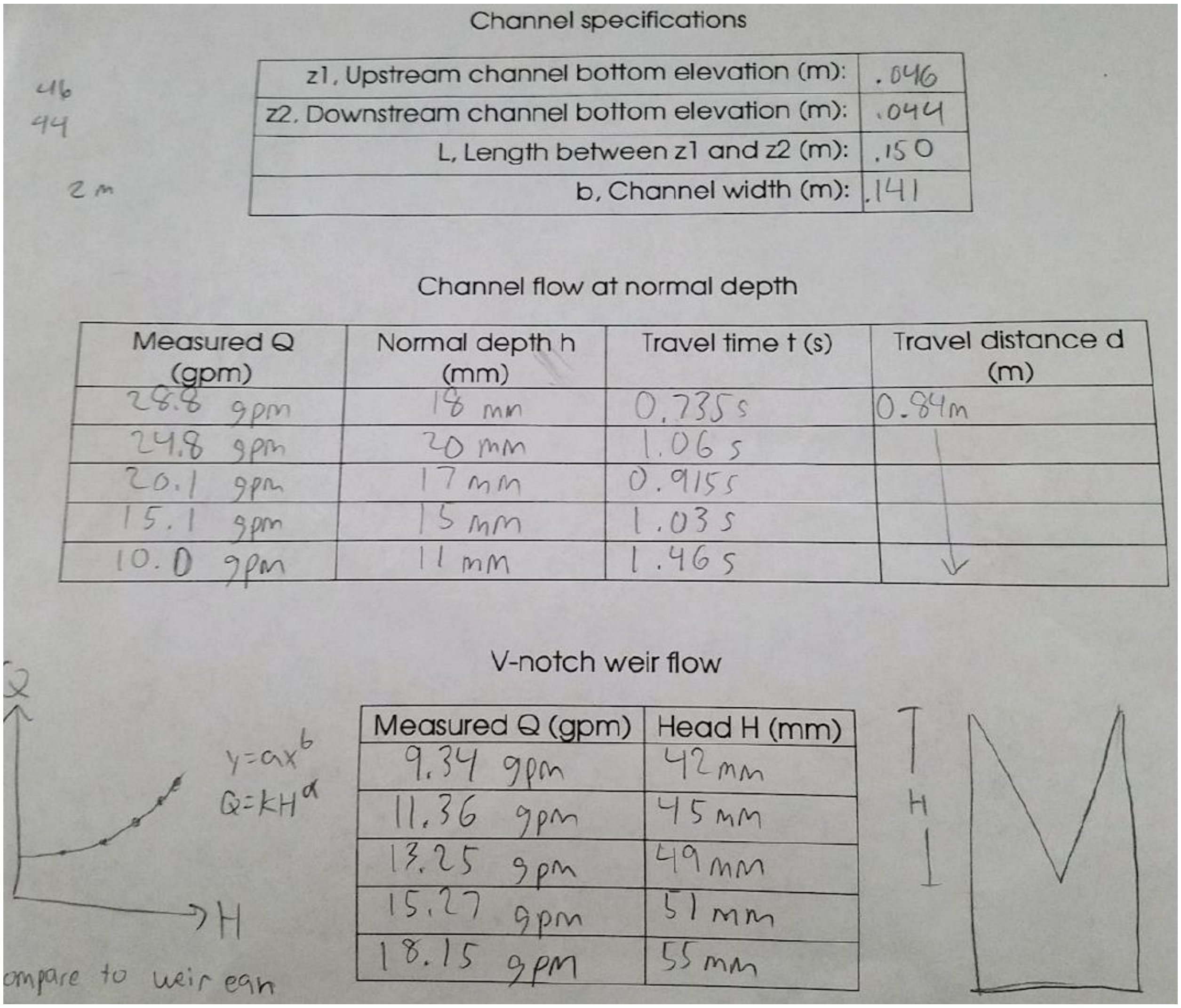
**Conclusion**

1. Recommend the best method for calculating flow rate in light of your measured data.

2. Recommend the best empirical equation to calculate flow rate over V-notch weirs.

**Sample Data Sheet**

Experiment 7: Open Channel Flow



Channel specifications

46
44
2 m

| | |
|---|---|
| z1, Upstream channel bottom elevation (m): | .046 |
| z2, Downstream channel bottom elevation (m): | .044 |
| L, Length between z1 and z2 (m): | .150 |
| b, Channel width (m): | .141 |

Channel flow at normal depth

| Measured Q (gpm) | Normal depth h (mm) | Travel time t (s) | Travel distance d (m) |
|---|---|---|---|
| 28.8 gpm | 18 mm | 0.735 s | 0.84 m |
| 24.8 gpm | 20 mm | 1.06 s | |
| 20.1 gpm | 17 mm | 0.915 s | |
| 15.1 gpm | 15 mm | 1.03 s | |
| 10.0 gpm | 11 mm | 1.46 s | |

V-notch weir flow

| Measured Q (gpm) | Head H (mm) |
|---|---|
| 9.34 gpm | 42 mm |
| 11.36 gpm | 45 mm |
| 13.25 gpm | 49 mm |
| 15.27 gpm | 51 mm |
| 18.15 gpm | 55 mm |

Q
$y=ax^b$
$Q=kH^a$
H
ompare to weir eqn
H

This page intentionally left blank.