THE HASHEMITE UNIVERSITY

Civil Engineering Department

110401358: Fluids and Hydraulic Lab

Instructor: Dr. Zeyad Tarawneh

Fall 2018 / 2019

Table of Contents

Exp. 1: Hydrostatic Force and Center of Pressure

Introduction and Theoretical Background

Civil engineering structures that retain water depth are subjected to a hydrostatic force, which occurs due to the water pressure on the structures surfaces. The design and stability analysis of the water retaining structures is a challenge to civil engineers, therefore the determination of the hydrostatic force magnitude and its point of application is essential. Fig. (1) shows the experimental setup that will be used to study the hydrostatic force.

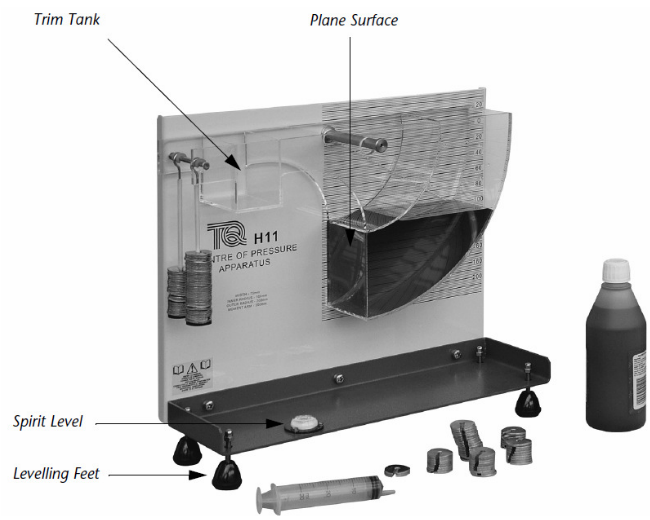

Fig. 1: The hydrostatic force experimental setup.

Fig. (2) below shows the hydrostatic pressure (p), hydrostatic force (F) and its point of application (the center of pressure: y_{cp}), the submerged surface inclination angle (α) from the horizontal and the water depth to an axis that passes through the centroid ($\bar{y}$) of the submerged surface.

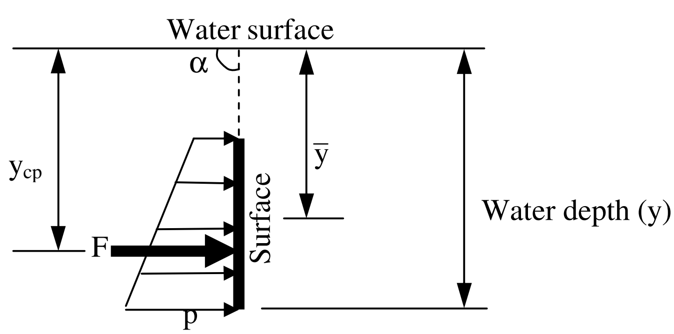

Fig. 2: The hydrostatic force (F) and its point of application (y_{cp}).

Theoretically, the magnitude of the hydrostatic force is F = $\overline{p} \times$ A, where $\overline{p}$ is the hydrostatic pressure at an axis that passes through the centroid of the submerged surface and A is the area of the submerged surface. Knowing the water depth at the centroid of the submerged surface ($\overline{y}$), then the force hydrostatic F is:

$$F = \rho g \times \overline{y} \sin\alpha \times A \qquad (1)$$

The depth (y_{cp}) at which the hydrostatic force applies on the submerged surface (the center of pressure) is:

$$y_{cp} = \overline{y} + \frac{\overline{I}}{\overline{y} \times A} \qquad (2)$$

where $\overline{I}$ is the moment of inertia about an axis that passes through the centroid of the submerged surface. The main objective of this experiment is to verify the theoretical computations of the hydrostatic force magnitude and its point of application (the center of pressure) for partially and fully immerged surfaces.

Case 1: partially submerged surface: water depth (y) < surface depth (d)

Referring to Fig. (3), taking the sum of moments about the pivot point (0) and knowing that F applies at (y/3) from the surface bottom, then: $F \times (R_2 - y/3) = mg \times R_3$. Therefore the experimental F is:

$$F_{exp} = mg \times R_3/(R_2 - y/3) \qquad (3)$$

where R_1 = 100mm, R_2 = 200mm and R_3 = 200mm.

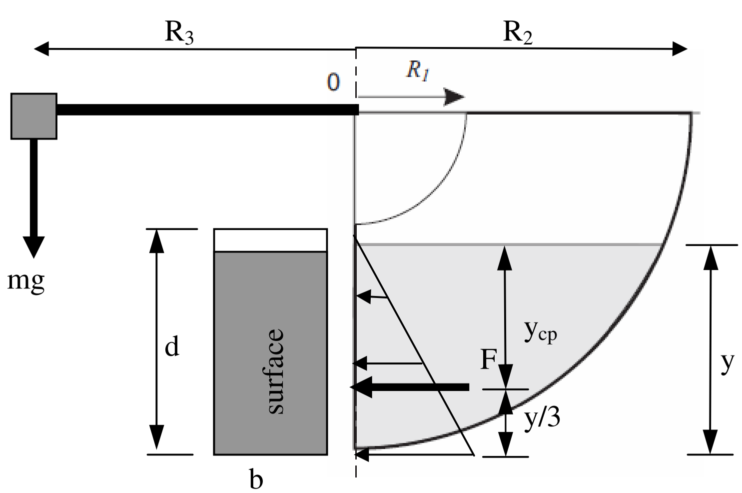

Fig. 3: Partially submerged surface of width (b = 75mm).

Similarly, to determine the experimental center of pressure ($y_{cp,\,exp}$), then given the force $F = \bar{p} \times A$ (noting that $\bar{p} = \rho g \times y/2$, A is the submergence area = b×y) and taking the sum of moment about the pivot point (0) then $F \times (R_2 - y + y_{cp}) = mg \times R_3$, and the experimental $y_{cp,\,exp}$ is:

$$y_{cp,exp} = \left(\frac{2m \times R_3}{\rho \times by^2} \right) - R_2 + y \qquad (4)$$

The experimental ($y_{cp,\,exp}$) is to be compared with the theoretical $y_{cp} = \bar{y} + \frac{\bar{I}}{\bar{y} \times A} = \frac{2}{3} y$ noting that $\bar{y} = y/2$, $\bar{I} = by^3/12$ and $A = by$.

Case 2: Fully submerged surface: water depth (y) > surface depth (d)

Referring to Fig. (4) and similar to the analysis conducted for the partially submerged surface, taking the sum of moment about (0), then the experimental F is:

$$F_{exp} = mg \times R_3 / (R_2 - [\frac{(yd/2) - (d^2/6)}{y - (d/2)}]) \qquad (5)$$

and the experimental $y_{cp,\,exp}$ is:

$$y_{cp,exp} = \left(\frac{2m \times R_3}{\rho \times (2ybd - bd^2)} \right) - R_2 + y \qquad (6)$$

The experimental ($y_{cp,\,exp}$) is to be compared with the theoretical $y_{cp} = \bar{y} + \frac{\bar{I}}{\bar{y} \times A}$ noting that $\bar{y} = y - (d/2)$, $\bar{I} = bd^3/12$ and $A = bd$.

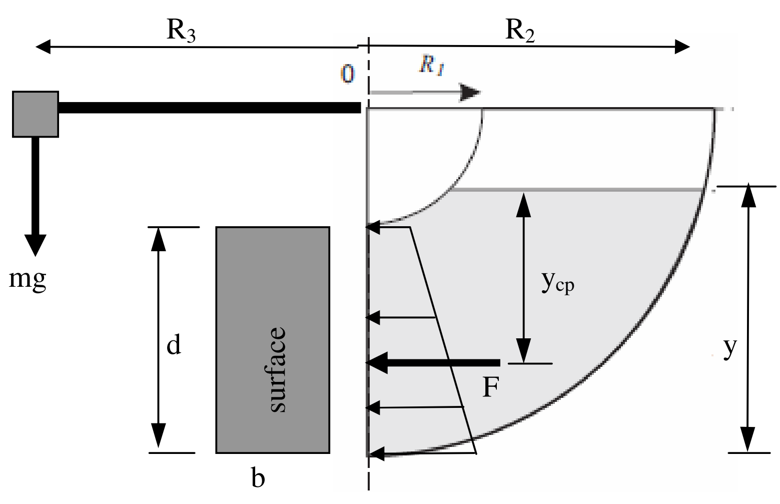

Fig. 4: Fully submerged surface of width (b = 75mm) and depth (d = 100mm).

Experiment Procedures

1. Hook an empty weight hanger to the support and add the colored water to the trim tank to create a balance such that the submerged surface is vertical.
2. Start with 10 grams mass. Place it in the hanger and add water to the quadrant tank until the tank becomes level. Record the water depth (y) versus the mass (m) for the partially submerged surface.
3. Repeat step 2 with 20 grams increments. Record the water depth (y) versus the mass (m) for the partially submerged surface.
4. Continue step 3 for fully submerged surface. Record the water depth (y) versus the mass (m).

Data

Refer to the lab data sheet if exists otherwise arrange your own tables as per instructions.

Computations and Results

1. Show at least one sample computation in your report or lab sheet.
2. For the initially and fully submerged surfaces, compute the theoretical and experimental F and plot F_{theo} versus F_{exp}.
3. For the initially and fully submerged surfaces, compute the theoretical and experimental y_{cp} and plot $y_{cp,\ theo}$ versus $y_{cp,\ exp}$.
4. Comment on results.

Exp. 2: Orifice and Jet Flow

Introduction and Theoretical Background

The orifice is a sudden contraction setup of circular shape (Fig. 1) that can be used to measure the actual flow rate leaving reservoirs, tanks or flow in pipes. When water flows through an orifice, the energy that causes the water flow is converted from one form to the other, which forms the basis of measuring the flow rate (Q) and the flow velocity (v).

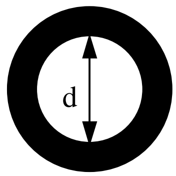

Fig. 1: Typical cross section in an orifice of diameter (d).

In the experiment, a small orifice of diameter d = 6mm is attached to small tank (Fig. 2). The water head (h) in the tank that causes the water flow will be kept constant during one measurement. The water flow leaves the orifice as trajectory of well known path, because it obeys the net force formed by the horizontal force due to the momentum when it leaves the orifice and the vertical gravitational force (downwards). At any point in the space, the trajectory path can be located after measuring the horizontal (X) and the vertical (Y) distances the flow travels. Such distances can be measured using needles, 10cm apart of each others, attached to the experimental setup (Fig. 2).

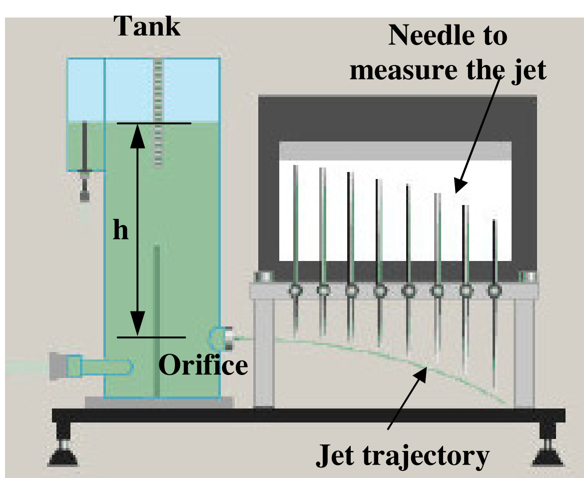

Fig. 2: Tank, orifice and jet trajectory.

Referring to Fig. (3), considering the dashed line passing through the orifice center as an energy reference line and applying the energy conservation principle between point 1 (the constant water surface in the tank) and point 2 (flow at orifice under the atmospheric pressure), then:

$$E_1 = E_2 + h_L \quad (1)$$

Equation (1) is reduced to:

$$h = v^2/2g + h_L \quad (2)$$

Given the measured water head (h), Equation (2) enables the computation of the flow velocity (v) if the head loss (h_L) is known. Unfortunately, h_L is unknown. It depends on the friction over the orifice (function of the orifice edge shape) and the flow actual velocity (function of the jet flow actual area).

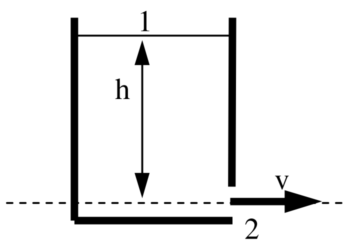

Fig. 3: Energy reference line and reference points.

In the meantime, if the h_L is ignored (later its effect will be introduced by a correction factor), then Equation (2) becomes:

$$v = \sqrt{2gh} \quad (3)$$

Equation (3) is valuable. It easily computes the flow theoretical velocity (v) by just knowing (h). Given the orifice diameter (d) then flow area (a) = $1/4\ \pi\ d^2$ and the theoretical flow rate ($Q = a \times v$) is:

$$Q = \frac{\pi}{4} d^2 \sqrt{2g}\sqrt{h} \quad (4)$$

Equation (4) computes the theoretical flow rate passing through an orifice as a function of the head (h) producing the water flow assuming that the head loss h_L is negligible, i.e. it gives the maximum possible Q. However, the h_L is always greater than zero, therefore the actual flow velocity is always less than the theoretical velocity and the actual flow rate (Q_a) is always less than the theoretical flow rate. Introducing the discharge coefficient (C_d) as the ratio of the actual flow rate to the theoretical flow rate that is always < 1:

$$C_d = Q_a / Q \quad (5)$$

The discharge coefficient (C_d) can be visualized as a correction factor to Equation (4) to compute the actual flow rate, therefore the actual flow (Q_a) is:

$$Q_a = C_d \frac{\pi}{4} d^2 \sqrt{2g}\sqrt{h} \quad (6)$$

Referring to Fig. (4), it should be known that the flow leaves the orifice as a jet of diameter (d_v) that is always less than the orifice diameter (d) due to the water viscosity. In Equation (2), the energy has been considered at point 2 (vena contracta) where the jet flow has diameter (d_v).

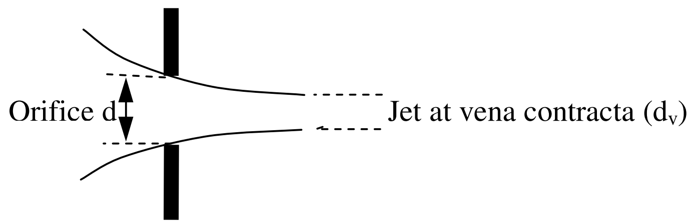

Fig. 4: Flow jet at vena contracta.

Therefore, to compute the actual jet velocity, Equation (3) has to be multiplied by a correction factor to compensate for the area reduction at the vena contracta. Introducing the velocity coefficient (C_v) as the ratio of the actual flow velocity to the theoretical velocity (C_v is always < 1), then:

$$C_v = \frac{v_a}{\sqrt{2gh}} \quad (7)$$

The main objectives of this experiment are to introduce the orifice as a flow rate measurement tool, obtain the orifice discharge coefficient (C_d) and the velocity coefficient (C_v).

The actual flow rate (Q_a) can be measured in the lab by collecting known water volume (V) in a given time (t), $Q_a = V / t$. Given that the flow jet is traveling horizontal distance (X) while falling vertical distance (Y) measured using the attached needles, then the actual flow velocity is:

$$v_a = \frac{X}{\sqrt{\frac{2Y}{g}}} \quad (8)$$

or,

$$C_v = \frac{X}{2\sqrt{Y h}} \quad (9)$$

Experiment Procedures

1. Fill the tank to a constant water depth (h). Record h.
2. Let water flows through the orifice and observe the jet trajectory as it leaves the orifice. Use needles to measure the jet horizontal and vertical distances (X and Y). Record X and Y.
3. Measure the water volume collected versus the time recorded. Record V and time.
4. Repeat the steps 1 – 3 at different h values.

Data

Refer to the lab data sheet if exists otherwise arrange your own tables as per instructions.

Computations and Results

1. Show at least one sample computation in your report or lab sheet.
2. Plot Q_a versus $\sqrt{h}$ and from the plot compute C_d.
3. Plot $Log(Q_a)$ versus Log(h) and determine the relation between Q_a and h.
4. Compute the velocity coefficient.
5. Comment on results.

Exp. 3: Bernoulli's Theorem and Venturi Meter

Introduction and Theoretical Background

The total energy for steady, incompressible and frictionless flow consists of three forms: potential energy (z) due to elevation, kinetic energy ($v^2/2g$) due to motion and energy due to pressure ($p/\rho g$). The total energy (E) expressed in units of meter (Joule / unit weight) is:

$$E = z + \frac{v^2}{2g} + \frac{p}{\rho g} \qquad (1)$$

In Equation (1), the pressure head ($p/\rho g$) is called the static head.

Fig. (1) below shows the experimental setup. It consists of a horizontal test tube connected with manometers at different points to measure the static head at these points and special sliding tube (Pitot tube) to measure the total head at any point.

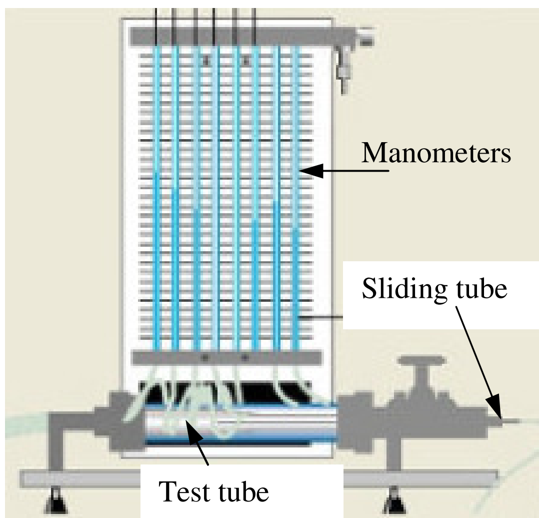

Fig. 1: The experimental setup.

The Bernoulli equation represents the conservation of the energy. It simply says: the total energy for steady, incompressible and frictionless flow in a system is conserved (constant) assuming no energy loss due to friction occurs, however the energy may convert from one form to the other. For example it may convert from pressure energy ($p/\rho g$) to kinetic energy ($v^2/2g$) and vice versa. To demonstrate that the total energy is conserved, the total head probe will be used. The total head probe or pitot tube (Fig. 2) consists of steel tube of stagnation point ($v = 0$) that slides horizontally along the venturi tube. At any point, the total head probe makes the kinetic energy head zero ($v^2/2g = 0$) thus that kinetic energy is converted to pressure head ($p/\rho g$) which can be measured using manometer #8.

If the total head probe is inserted through different points (1 – 7 along the horizontal venturi tube) then manometer #8 should give the same reading at all points assuming no friction loss occurs, i.e. the total energy is a pressure head because $v^2/2g = 0$, refer to Equation 1) which proves that the energy is conserved.

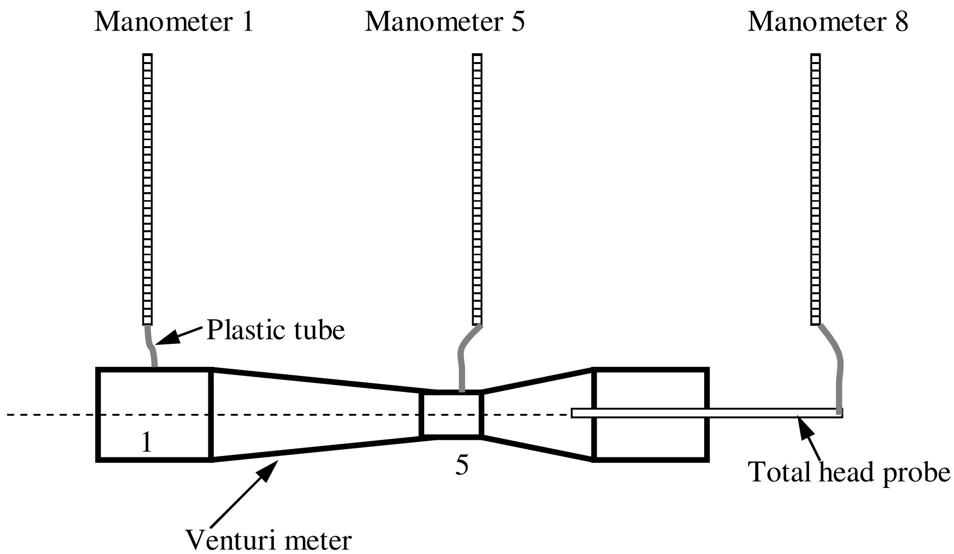

Fig. 2: Venturi tube, manometers and total head probe.

To measure the velocity head ($v_1^2/2g$) at point1 for example, the total head probe at point1 measures the total energy using manometer #8 while the local manometer (manometer1) measures the static head at point1 (the pressure head at point1 = $p_1/\rho g$). The difference between the total head and the static (pressure) head is the velocity head (kinetic part of the energy):

$$v_1^2/2g = \text{total head probe reading} - \text{static head reading} = H_{total} - p_1/\rho g \qquad (2)$$

Similar procedure can be used to measure the velocity head at any other points.

The venture meter is a gradual contraction setup (refer to Fig. 2) that can be used to measure the actual flow rate in pipes. When water flow passes through the venturi, the flow cross sectional area decreases due to the area contraction (between point1 of diameter d_1 = 25mm to point5 of diameter d_5 = 10mm). Since the flow rate (Q) is steady, i.e. conserved, then the velocity at the throat (point5) becomes higher than the velocity at point1. The increase in the velocity head at point5 is accompanied with reduction in the pressure head (static head reading at manometer5) because the total energy is conserved while one form of the energy converts to the other. Taking the energy conservation over points 1 and 5 in the horizontal test tube ($z_1 = z_5$) and assuming no head loss occurs due to friction and area contraction, then $E_1 = E_5$, or:

$$\frac{v_1^2}{2g} + \frac{p_1}{\rho g} = \frac{v_5^2}{2g} + \frac{p_5}{\rho g} \tag{3}$$

From the conservation of mass (continuity): $Q = A_1 \times v_1 = A_5 \times v_5$, then:

$$v_1 = \frac{A_5}{A_1} v_5 \tag{4}$$

Replacing $p_1/\rho g$ and $p_5/\rho g$ by h_1 and h_5 (the static head readings from the local manometers) and replacing Equation (4) in Equation (3), theN after simplification the theoretical flow rate is:

$$Q = A_5 \sqrt{\frac{2g(h_1 - h_5)}{1 - (A_5/A_1)^2}} \tag{5}$$

Equation (5) computes the theoretical flow rate passing through the venturi meter as a function of the pressure head before and after the throat (h_1 and h_5) assuming that the head loss h_L is neglected, i.e. the maximum possible Q. However, the h_L is always greater than zero, therefore the actual flow velocity is always less than the theoretical velocity and the actual flow rate (Q_a) is always less than the theoretical flow rate. Introducing the discharge coefficient (C_d) as the ratio of the actual flow rate to the theoretical flow rate that is always < 1, then the actual flow rate (Q_a) is:

$$Q_a = C_d A_5 \sqrt{\frac{2g(h_1 - h_5)}{1 - (A_5/A_1)^2}} \tag{6}$$

The main objectives of this experiment are to demonstrate that the energy is conserved, introduce the venturi meter as a water flow rate measurement tool and to obtain the orifice discharge coefficient (C_d). The actual flow rate (Q_a) can be measured in the lab by collecting known water volume (V) in a given time (t), $Q_a = V / t$.

Experiment Procedures

1. Let a stable flow runs through the test tube and wait until the manometers show stable readings.
2. Insert the total head probe at different points, observe and record the total head readings on manometer 8.
3. Remove the total head probe and read the static head at points 1 and 5.
4. Record the collected water volume (V) and time to obtain V.
5. Repeat the steps 1 – 4 at different Q values.

Data

Refer to the lab data sheet if exists otherwise arrange your own tables as per instructions.

Computations and Results

1. Show at least one sample computation in your report or lab sheet.
2. Plot Q_a versus $\sqrt{h_1 - h_5}$ and from the plot compute C_d.
3. Calculate C_d and Re # at each Q value and plot C_d versus Re #.
4. Comment on results.

Exp. 4: Impact of Water Jet

Introduction and Theoretical Background

Water in motion has momentum that is directly related to the velocity of the flow. The flow momentum if changed will create external forces on surfaces subjected to the water flow. As the energy and mass are conserved, the momentum is conserved as well. The momentum conservation principle says: the change in the momentum rate will appear as an external force in the direction where the momentum has changed. This external force is called the impact of water jet. For example, Fig. (1) shows the external forces a water block creates on a curved pipe line due to a change in momentum.

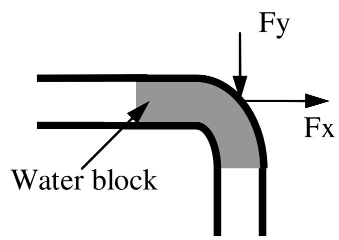

Fig. 1: External forces on the pipe surface due to the change in momentum.

In the experiment, the water flow from a pump will pass through a small nozzle (d = 8mm) that creates a vertical jet (Fig. 2). The vertical jet will hit an object (target of deflection angle either of 30°, 90°, 120° or 180°) and based on the deflection angle an external force applies on the target surface lifting the target up. The external force on the target can be equalized experimentally by an external weight (mass in the weight pan) upon which the target is balanced. Fig. (2) shows the experimental setup.

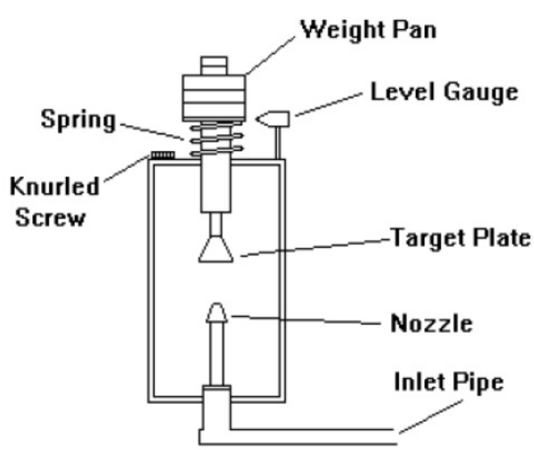

Fig. 2: Impact of water jet experimental setup.

Referring to Fig. (3), the water jet leaving the nozzle at velocity (v_n), hits the target at velocity (v_o) and later deflected by the target at angle θ from the vertical on both sides at velocity (v_1). For simplicity, it can be assumed that $v_1 = v_o$ under the assumption that the friction between the deflected jet and the target surface is negligible. Furthermore, applying the energy conservation between the nozzle and the location where the flow hits the target, then the velocity v_o can be computed using v_n as:

$$v_o^2 = v_n^2 - 2\,gS \qquad (1)$$

where S is the vertical distance between the nozzle and the target.

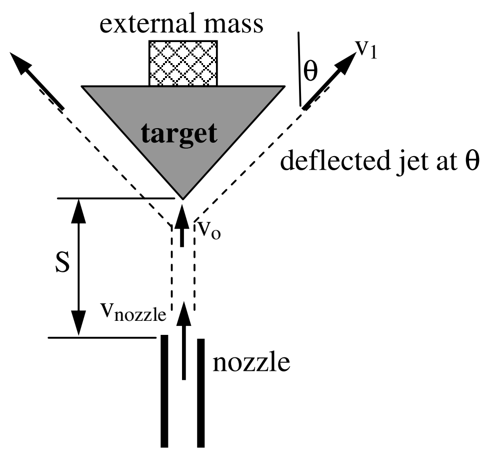

Fig. 3: Nozzle, target and deflected jet.

The momentum (M) is the product of the mass (m) by the velocity (v), M= m × v. The conservation of the momentum simply says:

$$\sum F = \frac{\Delta M}{\Delta t} \qquad (2)$$

From Equation (2), the vertical component of the force exerted on the water body due to the deflection and noting that $v_o = v_1$, is:

$$F = m \times v_1 \cos\theta - m \times v_o = m \times v_o (\cos\theta - 1) \qquad (3)$$

Based on Newton's 3rd law, the vertical force exerted by the deflected water jet on the target is:

$$F = m \times v_o (1 - \cos\theta) \qquad (4)$$

For fat plate, θ = 90, therefore from Equation (4), F = mv_o.

For hemisphere, θ = 180, therefore from Equation (4), F = $2mv_o$.

The rate of the quantity $m \times v_o$ in Equation (4) is called the mass flow rate that is equal to $\rho Q v_o$, where ρ is the water density. The force exerted b a deflected water jet is:

$$F = \rho Q v_o (1 - \cos\theta) \tag{5}$$

The main objectives of this experiment are to demonstrate the principle of momentum conservation and to measure the force exerted by a deflected water jet on a target.

Experiment Procedures

1. Fit the selected target on the setup and record the deflection angle θ.
2. Select an external balancing mass (m). Adjust the flow rate such that the target is balanced.
3. Record the mass (m), the collected water volume (V) and time to obtain V.
4. Repeat the steps 1 – 3 at different m and Q values.

Data

Refer to the lab data sheet if exists otherwise arrange your own tables as per instructions.

Computations and Results

1. Show at least one sample computation in your report or lab sheet.
2. Compute the experimental balancing force F = mg and the momentum $mv_o = \rho Q v_o$.
3. Plot F versus mv_o and determine the slope for each of the tested targets.
4. Comment on results.

Exp. 5: Friction in Pipes and Losses from Fittings and Bends

Introduction and Theoretical Background

When water flows in a pipe of length (L), the total energy overall the pipe is conserved. However, part of the energy will be converted to head loss (probably heat due to friction) to overcome the internal roughness of the pipe and/or the viscosity of the water (water has the ability to stick on the pipe inside wall). Fig. (1) shows a pipe of length (L), diameter (D) that discharges flow rate of velocity (v) from point 1 to point 2, energy components including head loss (h_L), energy grade line (EGL) and hydraulic grade line (HGL).

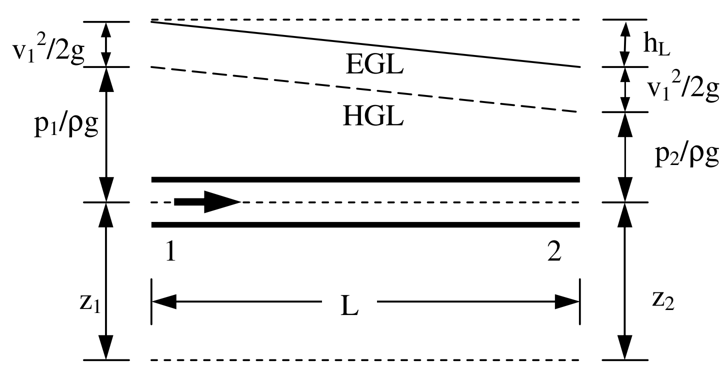

Fig. 1: Energy components over a pipe of length (L).

Referring to Fig. (1), taking the energy between points 1 and 2 over the horizontal pipe ($z_1 = z_2$) assuming the flow in the pipe is steady and incompressible then:

$$E_1 = E_2 + h_L \tag{1}$$

Replacing the energy components and noting that $z_1 = z_2$ and $v_1 = v_2$ (uniform flow of the same area over the whole pipe because the pipe diameter is constant), then:

$$\frac{p_1}{\rho g} = \frac{p_2}{\rho g} + h_L \tag{2}$$

The term $p_1/\rho g = h_1$ is the static (pressure) head at point 1 which can be measured easily using the manometer, similarly $p_2/\rho g = h_2$. Therefore, the experimental $h_L = h_1 - h_2$.

The Darcy equation is used to compute the flow velocity (v) as a function of the energy difference between points 1 and 2. The theoretical flow velocity is:

$$v = \sqrt{8g/f}\sqrt{R}\sqrt{S_f} \tag{3}$$

Where f is the Darcy friction factor (function of the flow Re # and the pipe relative roughness), R is the flow hydraulic radius = flow area / wetted perimeter (R = A/P) and S_f is the slope of the energy grade line (the energy difference over the pipe that causes the flow motion). Referring to Fig. 1, the slope of the EGL is $S_f = h_L/L$, and $R = (1/4)\pi D^2 / \pi D = D/4$. In Equation (3), replacing S_f by h_L/L and R by D/4 and simplifying then the theoretical energy friction loss (h_L) is:

$$h_L = \frac{f \times L}{D}\frac{v^2}{2g} \qquad (4)$$

If the pipe internal surface is extremely smooth (relatively no internal roughness) and the flow has a very low velocity then the flow is said to be nearly laminar. In that case the Darcy friction factor (f) is function of the Re # only, i.e. the water viscosity only. For the laminar flow, f = 64 / Re, and the $Re = \rho v D/\mu$ (μ is the dynamic viscosity that can be obtained from tables as a function of the water temperature). Replacing f by 64/Re in Equation (4), then the theoretical h_L for the laminar flow is:

$$h_L = \frac{32\mu L v}{\rho g D^2} \qquad (5)$$

If the pipe internal surface is rough and the flow has high velocity then the flow is said to be turbulent and in that case the Darcy friction factor (f) is function of the Re # (water viscosity) and the pipe relative roughness. The h_L for the turbulent flow is given by Equation (4). Referring to Equations (4 and 5), it can be seen that the $h_L \alpha v^2$ for the turbulent flow while $h_L \alpha v$ for the laminar flow.

The general form of the energy loss (h_L) from pipe bends or fittings (valves for example) is:

$$h_L = K\frac{v^2}{2g} \qquad (6)$$

where K is the energy loss factor that depends on the bend angle and radius of bending or the nominal size of the valve and its relative opening (for example 1/2 open, 3/4 open and so on).

Fig. (2) below shows the experiment setup. It consists of horizontal straight pipes of different materials and diameters, different types of valves (globe valve, gate valve, etc.) and different types of bends (smooth bend, sharp bend, etc.)

Fig. 2: The experimental setup.

The main objectives of this experiment are to determine the relation between the energy head loss (h_L) and the flow velocity (v) for laminar (if possible) and turbulent flows, obtain the Darcy friction factor (f) for pipes and the energy loss factor (K) for fittings and bends.

Experiment Procedures

1. Let water runs through the pipe selected to test and record the pipe diameter (d). Read and record the static head over both ends of the pipe and record the water volume and time to compute Q and v. Repeat step 1 at different Q and v values.
2. Select a bend to test and let water runs. Read and record the static head over both ends of the bend and record the water volume and time to compute Q and v given the pipe diameter. Repeat step 2 at different Q and v values.
3. Repeat step 2 for a selected fitting.

Data

Refer to the lab data sheet if exists otherwise arrange your own tables as per instructions.

Computations and Results

1. Show at least one sample computation in your report or lab sheet.
2. Compute the experimental h_L and v for the tested pipe and plot h_L versus v. Determine if the flow is laminar of turbulent. Also, compute f and Re and plot f versus Re.
3. Use the experimental h_L and v for the tested fittings and bends to compute K.
4. Comment on results.

Exp. 6: Specific Energy: Slow and Fast Flow

Introduction and Theoretical Background

The total energy concept and equation illustrated in Experiment (3) will be used to define the specific energy. In open channels, the specific energy is defined as the total energy measured at a point lies on the channel bed (the energy reference line is taken as the channel bed). Fig. (1) shows the components of the specific energy and the side view of a channel that discharges flow of velocity (v) and depth (h).

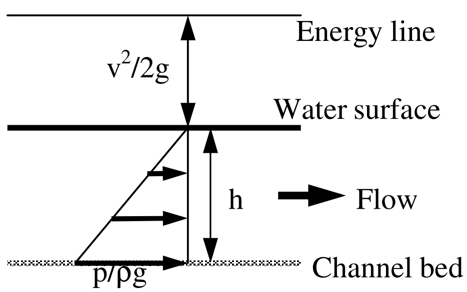

Fig. 1: Specific energy components for open channel flow of velocity (v) and depth (h).

The total energy (E) expressed in units of meter for a point that lies on the channel bed is:

$$E = z + \frac{v^2}{2g} + \frac{p}{\rho g} \tag{1}$$

Referring to Fig. (1), the static pressure at that point is the hydrostatic pressure, thus $p/\rho g = h$ (the flow depth) and noting that the point under consideration lies on the energy reference line, i.e $z = 0$, then the total energy described by Equation (1) is the specific energy that becomes:

$$E = h + \frac{v^2}{2g} \tag{2}$$

In Equation (2), replacing the flow velocity (v) by Q/A, where A is the flow cross sectional area and Q is the steady flow, then:

$$E = h + \frac{Q^2}{2gA^2} \tag{3}$$

Noting that A is function of the flow depth (h) and Q is constant, then Equation (3) describes (relates) the specific energy (E) versus the flow depth (h). If E is plotted against h then a unique relationship is observed as demonstrated by Fig. 2. The specific energy takes a minimum value (E_{min}) when the flow is critical (the flow depth h = the critical depth h_c). When $h > h_c$, then the flow is subcritical (slow stream $v < v_c$), and when $h < h_c$, then the flow is supercritical (fast stream $v > v_c$).

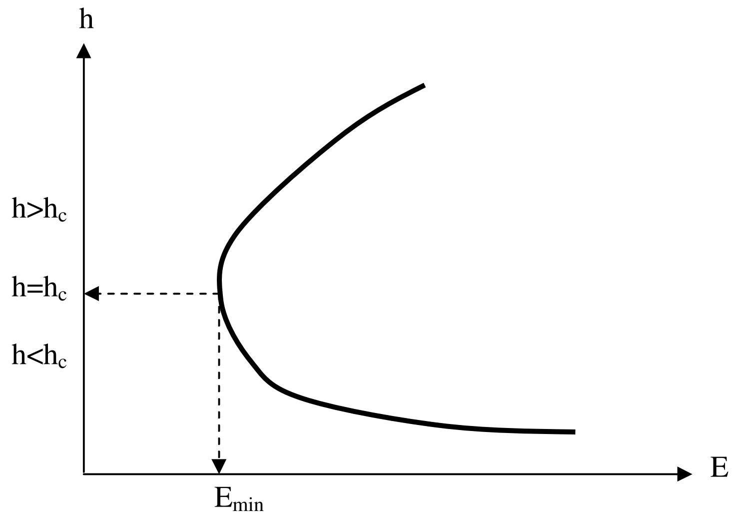

Fig. 2: The specific energy (E) versus the flow depth (h).

Since E takes a minimum value at the critical condition, then deriving E with respect to the only variable h and observing that dE/dh = 0 (at the minimum value), then the general form of the Froude# (Fr) is:

$$Fr^2 = \frac{Q^2 T}{gA^3} \quad (4)$$

where T is the flow top surface width. The Fr = 1 for the critical flow, >1 for supercritical and < 1 for the subcritical flow. From Equation (4), the reduced form of the Fr # becomes:

$$Fr = \frac{v}{\sqrt{gh_m}} \quad (5)$$

where h_m is the mean hydraulic depth = A / T. For rectangular flow cross sectional area in rectangular channels, then T = b (the bottom width) and the critical depth (h_c) becomes:

$$h_c = \left(\frac{q^2}{g}\right)^{1/3} \quad (6)$$

where q is the distributed flow rate, i.e. q = Q / b. Also, it can be shown that the theoretical E_{min} for rectangular section flow, i.e. $E_{min} = (3/2)\ h_c$.

Fig. 3 below shows the experiment setup. It consists of horizontal rectangular section channel of 75mm width, sluice gate to regulate the flow, gauges to measure the flow depth.

Fig. 3: The experimental setup.

The main objectives of this experiment are to determine the relation between the specific energy and the flow depth (h), determine the flow critical depth and minimum energy and distinguish between flow types.

Experiment Procedures

1. Select a flow rate and let water runs in the channel. Record the flow rate.
2. Adjust the sluice gate opening to regulate the flow on both sides of the gate. Record the flow depths (h) upstream and downstream the gate.
3. Repeat step 2 at different gate openings. Record the flow depths upstream and downstream the gate. The experiment requires at least 6 - 8 depth values upstream and downstream the gate.

Data

Refer to the lab data sheet if exists otherwise arrange your own tables as per instructions.

Computations and Results

1. Show at least one sample computation in your report or lab sheet.
2. Compute the specific energy E at the different flow depths and plot E (on the x-axis) versus h (on the y-axis). Determine the experimental critical depth and compare with the theoretical h_c.
3. Use the plot to determine the experimental E_{min} and compare it with the theoretical E_{min}.
4. Comment on results.

Exp. 7: Hydraulic Jump

Introduction and Theoretical Background

The hydraulic jump is a natural phenomenon of turbulence occurs when the open channel flow is forced to change its status from supercritical (fast stream) to subcritical (slow stream). Over the jump, a huge reduction in the flow velocity occurs, i.e. before the jump the flow has high velocity and low flow depth while after the jump the flow has low velocity accompanied with high depth, therefore, the flow depth has jumped from low to high value. In the jump turbulence, the flow stream paths will strongly collide with each other causing a considerable reduction in the flow velocity and a huge loss in the energy. Fig. 1 shows the development of the hydraulic jump between point 1 (before) and point 2 (after).

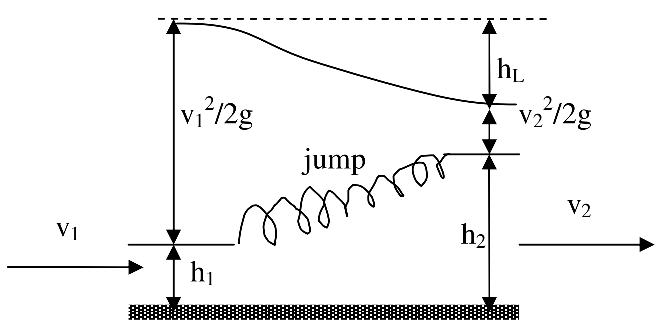

Fig. 1: The hydraulic jump.

The energy (E) conservation between points 1 and 2 is:

$$E_1 = E_2 + h_L \qquad (1)$$

Equation (1) above can't be used to compute the flow depth before or after the jump because the h_L is unknown for most of channel sections, except the rectangular section, i.e. the energy equation although correct but useless (it has 2 unknowns: the flow depth and h_L). As an alternative procedure, the momentum conservation is used. The momentum has no loss and any change in the momentum will appear as external forces that can be measured or computed easily. Applying the momentum conservation before and after the jump, then:

$$\bar{p}_1 A_1 + \rho Q v_1 = \bar{p}_2 A_2 + \rho Q v_2 \qquad (2)$$

Where $\bar{p}_1$ and $\bar{p}_2$ is the hydrostatic pressure at the centroid of the flow area A_1 and A_2 respectively. For flow of rectangular section, it can be shown that Equation (2) is reduced to:

$$h_2 = \frac{h_1}{2}\left(\sqrt{1+8Fr_1^2} - 1\right) \qquad (3)$$

or:

$$h_1 = \frac{h_2}{2}\left(\sqrt{1+8\,Fr_2^2} - 1\right) \tag{4}$$

Given h_1 and h_2, then the energy loss can be calculated using the energy equation as:

$$h_L = E_1 - E_2 \tag{5}$$

For flow of rectangular section, Equation (5) above becomes:

$$h_L = \frac{(h_2 - h_1)^3}{4h_1h_2} \tag{6}$$

The height of the jump is calculated as $Hj = h_2 - h_1$.

Theoretically, the length of the jump can't be computed using close form equations, it can be measured experimentally. Table 8.5 in "Understanding Hydraulics by Less Hamill, 2nd edition" gives experimental values for the jump length as a function of the Fr_1.

Fig. 2 below shows the experiment setup. It consists of horizontal rectangular section channel of 75mm width, sluice gate to regulate the flow, gauges to measure the flow depth.

Fig. 2: The experimental setup.

The main objectives of this experiment are to observe the development of the hydraulic jump, compute the flow depth after the jump and energy loss over the jump.

Experiment Procedures

1. Select a flow rate and let water runs in the channel. Record the flow rate.
2. Use the sluice gate and the channel end gate to regulate the flow and develop the jump.
3. Record the flow depth upstream and downstream the jump. Record the jump length.
4. Repeat the steps above at different flow rates.

Data

Refer to the lab data sheet if exists otherwise arrange your own tables as per instructions.

Computations and Results

1. Show at least one sample computation in your report or lab sheet.
2. Given the measured flow depth before the jump, compute the theoretical flow depth after the jump and compare with the measured value. Given the measured flow depths, compute the experimental energy loss and compare with theoretical value.
3. Compute the jump height and compare the experimental jump length versus the theoretical value.
4. Comment on results.

Exp. 8: The Flow Beneath a Sluice Gate (An Undershot Weir)

Introduction and Theoretical Background

The sluice gate is a tool that can be used to control and regulate the open channel flow. In addition, it can be used to measure the flow rate when acting as an undershot weir (inverted weir). Fig. 1 below shows the sluice gate, the opening under the gate (h_g) and flow characteristics at points 0 and 1, before and after the gate respectively.

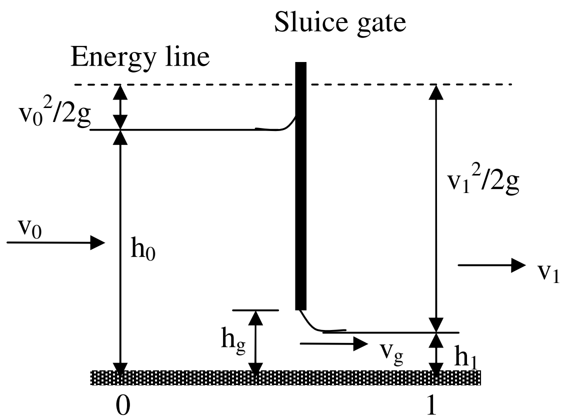

Fig. 1: The sluice gate and flow characteristics up and downstream the gate.

Taking the energy (E) between points 0 and 1, then $E_0 = E_1 + h_L$. After neglecting the energy head loss:

$$h_0 + v_0^2/2g = h_1 + v_1^2/2g \quad (1)$$

The flow upstream the gate is subcritical and v_0 is actually very low, therefore the term $v_0^2/2g$ can be assumed zero. Furthermore, the flow depth downstream the gate (h_1) is also very low (supercritical flow) which can be neglected under the fact that the majority of the energy after the gate is due to the velocity head ($v_1^2/2g$) only. Given that the velocity head $v_1^2/2g \approx v_g^2/2g$, then Equation (1) reduces to:

$$v_g = \sqrt{2g \times h_0} \quad (2)$$

Equation (2) gives the theoretical flow velocity as it passes under the gate. Given the flow area ($b \times h_g$) then the theoretical flow rate is:

$$Q = bh_g\sqrt{2g \times h_0} \quad (3)$$

Defining the discharge coefficient ($C_d = Q_a/Q$) as previously, then the actual flow rate is:

$$Q_a = C_d b h_g \sqrt{2g \times h_0} \quad (4)$$

The main objectives of this experiment are to introduce the sluice gate as flow regulation tool, compute the flow rate and the discharge coefficient.

Experiment Procedures

1. Open the gate at h_g = 15mm. Start with low flow rate such that h_0 is always > h_g. Let the water runs, measure and record experimental Q_a and h_0.
2. At the same gate opening (h_g = 15mm), increase the flow rate and record Q_a and h_0.
3. Repeat the steps above for h_g = 25mm.

Data

Refer to the lab data sheet if exists otherwise arrange your own tables as per instructions.

Computations and Results

1. Show at least one sample computation in your report or lab sheet.
2. For each h_g listed above, plot Q_a versus $\sqrt{h_0}$ and from the plot determine sluice gate C_d at each h_g opening.
3. Plot C_d versus h_g.
4. Comment on results.

Exp. 9: The Flow Through Rectangular Sharp Crested Weir

Introduction and Theoretical Background

Sharp and broad crested weirs are tools used to measure the open channel flow. In general, the flow rate in small channels is measured using sharp crested weirs while the flow in large channels is measured using the broad crested weir. Fig. 1 below shows a side view of the flow as it passes through rectangular sharp crested weir of width (b).

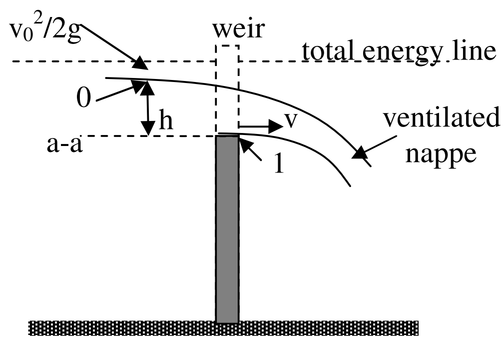

Fig. 1: Side view of weir, flow just upstream the weir, and ventilated nappe.

Similar to the analysis of orifice, taking the total energy (E), with respect to the line a-a that passes at the weir crest, between point 0 (on the water surface just upstream the weir) and point 1 (at the weir crest) and noting that the ventilated nappe is exposed to the atmospheric pressure, then $E_0 = E_1 + h_L$. After neglecting the energy head loss and noting that $z_0 = h$, $z_1 = 0$, $p_0/\rho g = p_1/\rho g = 0$ (atmospheric pressure) and the flow upstream the weir is subcritical ($v_0^2/2g \approx 0$ because v_0 is very low) then:

$$v = \sqrt{2g}\ h^{1/2} \tag{1}$$

Equation (1) measures the velocity at the nappe bottom (exactly at weir crest) noting that v varies according to the location over the line a-a (over the entire nappe depth). Therefore to use Equation (1) to compute the flow rate over the entire nappe depth, it is necessary to define incremental elements each of (dh) depth and dQ flow and integrate all over the entire depth (h) to estimate the total Q passes through weir as can be seen from Fig. 2 below. The incremental flow dQ = v×dA, where dA = b×dh. Integrating dQ, then the theoretical flow rate Q is:

$$Q = \int v dA = b\sqrt{2g}\int_0^H h^{1/2} dh = \frac{2}{3} b\sqrt{2g}\, H^{3/2} \tag{2}$$

The actual flow rate is obtained given the discharge coefficient C_d as:

$$Q_a = \frac{2}{3} C_d b\sqrt{2g}\, H^{3/2} \tag{3}$$

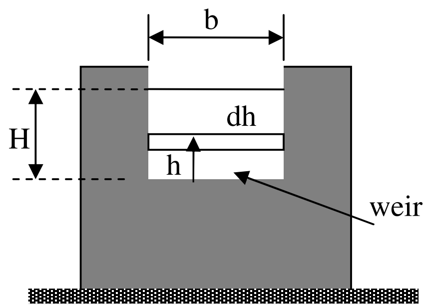

Fig. 2: Cross sectional view of the weir of width (b) and the incremental element (dh)

The main objectives of this experiment are to introduce the rectangular sharp crested weir as flow measurement tool in small channels, compute the flow rate and estimate the weir discharge coefficient.

Experiment Procedures

1. Place the sharp weir in the channel and start with low flow rate. Let the water runs, record Q_a and H. Make sure that the nappe after the weir is ventilated.
2. Increase the flow rate and record the new Q_a and H values. Make sure that the nappe after the weir is ventilated.

Data

Refer to the lab data sheet if exists otherwise arrange your own tables as per instructions.

Computations and Results

1. Show at least one sample computation in your report or lab sheet.
2. Plot Q_a versus $H^{3/2}$ and from the plot determine the sharp weir C_d.
3. Plot logQ_a versus logH and determine the power exponent (n) of the relation $Q \alpha H^n$.
4. At each Q_a, compute C_d and plot C_d versus H.
5. Comment on results.

Exp. 10: Pumps in Parallel and Series

Introduction and Theoretical Background

Pumps are hydraulic machines that convert the traditional power (electrical or mechanical) to a hydraulic power. The duty required by a pump is submit the targeted flow rate (Q) to the targeted total head (H) which are the components of the hydraulic power (P_{ow}). The hydraulic power in units of watt is:

$$P_{ow} = \rho g Q H \tag{1}$$

Equation (1) describes the two loads (Q, H) that affect the pump performance. The relationship between the pump two loads is unique and inversely proportional, i.e. as the flow rate Q is increased the delivered head (H) decreases. Since the pump duty is to convert the traditional power to a hydraulic power then the pump efficiency (ε) can be computed as:

$$\varepsilon = \frac{\text{output power}}{\text{input power}} = \frac{\rho g Q H}{2\pi T\,(N/60)} \tag{2}$$

where T is the torque generated (N.m) and N is the pump rotating speed (rpm).

In this experiment, the performance of a single pump, two identical pumps in parallel and series will be studied. Fig. 1 below shows the experimental setup.

Fig. 1: The experimental setup: pumps and connections.

Referring to Fig. (2), when the single pump fails to overcome the targeted total head, then two (or more) identical pumps are connected in series to magnify the resulted head. In that case the total flow rate submitted equals the single pump flow rate while the total head delivered is addition of the single pump total head.

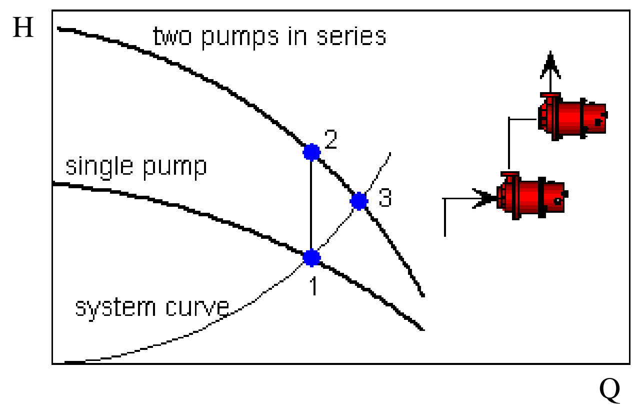

Fig. 2: The performance of two pumps in series versus the single pump performance.

Referring to Fig. (3), when the single pump fails to overcome the targeted flow rate, then two (or more) identical pumps are connected in parallel to magnify the resulted flow rate. In that case the total head delivered equals the single pump total head while the total flow rate submitted is the addition of the single pumps flow rate.

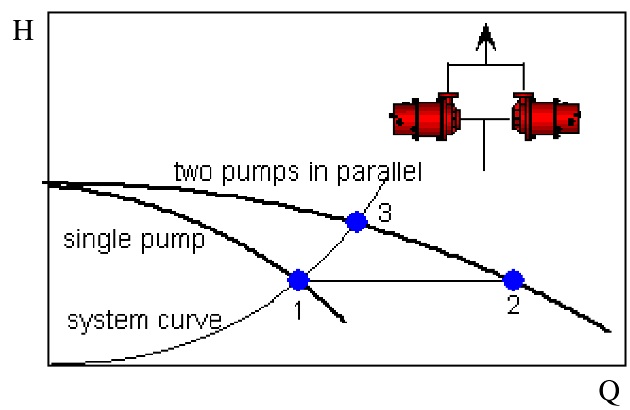

Fig. 3: The performance of two pumps in series versus the single pump performance.

The main objectives of this experiment are to determine the performance of the single pump, two pumps in series, two pumps in parallel and to compute the single pump efficiency.

Experiment Procedures

1. For the single pump, make arrangements to use one pump. Start with low flow rate, record the pump suction and delivery heads, the pump rotational speed (N) and torque (T) generated. Repeat this step for other two different flow rates.

2. For two pumps in parallel, make arrangements to connect the two pumps in parallel. Start with low flow rate, record the connected pumps suction and delivery heads. Repeat this step for other two different flow rates.
3. For two pumps in series, make arrangements to connect the two pumps in series. Start with low flow rate, record the connected pumps suction and delivery heads. Repeat this step for other two different flow rates.

Data

Refer to the lab data sheet if exists otherwise arrange your own tables as per instructions.

Computations and Results

1. Show at least one sample computation in your report or lab sheet.
2. Plot H versus Q for the single pump. Compute ε and plot it on the H-Q pump performance plot.
3. On the same H-Q plot. Plot the H-Q relation for two pumps in series and in parallel.
4. Comment on results.