Ethiopian TVET-System

Electro Mechanical Equipment Maintenance Supervision

Level IV

Based on March, 2017G.C. Occupational Standard

Module Title: Apply Principles of Hydraulics to Pipe and Channel Flow

TTLM Code: EIS EES4 TTLM 0820 v1

Aug, 2020

This module includes the following Learning Guides

LG17: Calculate energy losses in pipe flow

LG Code: EIS EES4 M09LO1-LG-42

LG18: Calculate hydraulic and energy gradient for pipelines

LG Code: EIS EES4 M09LO2-LG-43

LG19: Calculate flow in open channels

LG Code: EIS EES4 M09LO3-LG-44

LG20: Calculate flows through notches and weirs

LG Code: EIS EES4 M09LO4-LG-45

LG20: Calculate proportions for an economic section

LG Code: EIS EES4 M09LO5-LG-46

Instruction Sheet	Learning Guide 42: Calculate energy losses in pipe flow

This learning guide is developed to provide you the necessary information regarding the following **content coverage** and topics:

- Energy losses in pipe flow
- hydraulic Measurements
- Applying hydraulic *software*
- Identifying inconsistent data on *flow conditions*.

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, **upon completion of this Learning Guide, you will be able to**:

- .Measurements are reviewed and compared against expected trends.
- Standard processes and software are used to check, edit, verify and audit data.
- Standard processes are used to identify, estimate, adjust and justify data and review inconsistent data on flow conditions.
- Records are prepared in a format suitable for dissemination

Learning Instructions:

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below
3. Read the information written in the "Information Sheets 1- 4". Try to understand what are being discussed.
4. Accomplish the "Self-checks1,2,3 and 4 " in each information sheets on pages 16,21,31and 34.
5. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-checks).
6. After You accomplish self check, ensure you have a formative assessment and get a satisfactory result; then proceed to the next LG.

Information Sheet-1	Energy losses in pipe flow

Introduction

The total energy loss in a pipe system is the sum of the major and minor losses. Major losses are associated with frictional energy loss that is caused by the viscous effects of the fluid and roughness of the pipe wall. Major losses create a pressure drop along the pipe since the pressure must work to overcome the frictional resistance. The Darcy-Weisbach equation is the most widely accepted formula for determining the energy loss in pipe flow. In this equation, the friction factor (f), a dimensionless quantity, is used to describe the friction loss in a pipe. In laminar flows, f is only a function of the Reynolds number and is independent of the surface roughness of the pipe. In fully turbulent flows, f depends on both the Reynolds number and relative roughness of the pipe wall.

1.1 Calculating energy losses in pipe flows

Our intension here is generalized the one-dimensional Bernoulli equation for viscous flow. When the viscosity of the fluid is taken into account total energy head $H= V^2/2g + p/\rho g + z$ is no longer constant along the pipe. In direction of flow, due to friction cause by viscosity of the fluid we have $V1^2/2g+p1/\rho g+z1>V2^2/2g+p2/\rho g+z2$. So to restore the equality we must add some scalar quantity to the right side of this inequality

$$\boxed{\frac{v_1^2}{2g}+\frac{p_1}{\rho g}+z_1=\frac{v_2^2}{2g}+\frac{p_2}{\rho g}+z_2+\Delta h_{ls}} \text{ ------------------------ (1)}$$

This scalar quantity Δ_{ls} is called as hydraulic loss. The hydraulic loss between two different cross section along the pipe is equal to the difference of total energy for this cross section:

$$\boxed{\Delta h_{ls}=H_1-H_2} \text{ --- (2)}$$

We must remember that always $H_1>H_2$. In horizontal pipe when $z_1=z_2$ and diameter of pipe is constant $v_1=v_2$ hydraulic loss is equal to the head of pressure drop or head loss

$$\Delta h_L=\frac{p_1-p_2}{\rho g} \text{ ------------------------------------- (3)}$$

Head loss is express by Darcy -Weisbach equation:

$$\boxed{h_L=f\,\frac{L}{D}\,\frac{v^2}{2g}} \text{ ------------------------------ (4)}$$

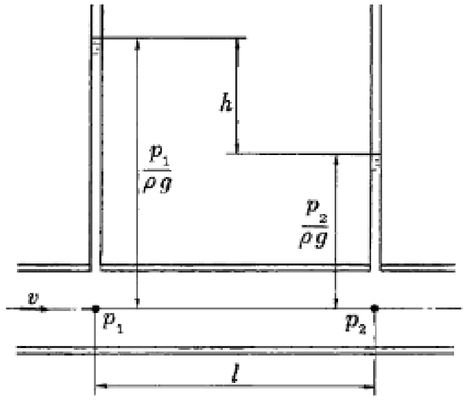


Figure 1: Pipe friction loss. For horizontal pipe, with constant diameter this loss may be measured by height of the pressure drop: $\frac{\Delta p}{\rho g} = h$

Figure 1.1 pipe friction loss

We must remember that equation (4) is valid only for horizontal pipes. In general, with v1=v2 but z1‡z2, the head loss is given

$$\frac{p_1 - p_2}{\rho g} = (z_2 - z_1) + f\,\frac{L}{D}\,\frac{v^2}{2g}$$ ----------------------------------- (5)

Part of the pressure change is due to elevation change and part is due to head loss associated with frictional effects, which are given in terms of thefriction factor fthat depends on Reynolds number and relative roughness f=φ(Re,ε/D).It is not easy to determine the functional dependence of the friction factor on the Reynoldsnumber and relative roughness(ε/D). Nikuradse used artificially roughened pipes produced by gluing sand grains of known size onto pipe walls to produce pipes with sand paper type surfaces. In commercially available pipes the roughness isnot as uniform and well defined as in the artificially roughened pipes used by Nikuradse. However, it is possible to obtain a measure of the effective relative roughness of typical pipes and thus to obtain the friction factor. Figure (3)) shows the functional dependence off on Reand and is called the Moody chart in honor of L. F.Moody, who, along with C. F. Colebrook, correlated the original data of Nikuradse in terms of the relative roughness of commercially available pipe materials.

Types of Fluid Flow Problems

In the design and analysis of piping systems that involve the use of the Moody chart, we usually encounter three types of problems:

1. Determining the pressure drop when the pipe length and diameter are given for a specified flow rate (or velocity).

Example 1. *Oil, with* $\rho = 900kg/m3$ *and kinematic coefficient of viscosity* $\nu = 0,00001m2/s$, *flows at* $q_v = 0,2m3/s$ *through 500 m of 200-mm diameter cast-iron pipe. Determine (a) the head loss and (b) the pressure drop if the pipe slopes down at 10 in the flow direction.*

Solution. *First we compute the Reynolds number* $Re = V \cdot d/\nu = 4 \cdot q_v/(\pi d\nu) = 4 \cdot 0,2/(3,14 \cdot 0,2 \cdot 0,00001) = 128000$. *Velocity is equal to* $v = q_v/(\frac{\pi d^2}{4}) = 6.4\ m/s$. *Absolute roughness for iron-cast pipe is* $\varepsilon = 0.26\ mm$. *So relative roughness is* $\varepsilon = \frac{\varepsilon}{d} = 0,0013$. *Now we are able to calculate using the formula(10) or Moody chart, friction factor* $f = 0,0225$. *Then the head loss is*

$$h_f = f\frac{L}{d}\frac{v^2}{2g} = 0,0225\frac{500}{0.2}\frac{6,4^2}{2 \cdot 9,81} = 117\ m$$

For inclined pipe the head loss is

$$h_f = \frac{\Delta p}{\rho g} + z_1 - z_2 = \frac{\Delta p}{\rho g} + L\sin 10^o.$$

So pressure drop is

$$\Delta p = \rho g(h_f - 500 \cdot \sin 10^o) = 900 \cdot 9,81 \cdot (117 - 87) = 265 \cdot 10^3.$$

2. Determining the flow rate when the pipe length and diameter are given for a specified pressure drop

Example 2. *Oil, with* $\rho = 950\ kg/m3$ *and* $\neq = 2E-5\ m2/s$, *flows through a* $30-cm$*-diameter pipe 100 m long with a head loss of 8 m. The roughness ratio is* $\varepsilon/d = 0.0002$. *Find the average velocity and flow rate.*

Iterative Solution. *To start we need to guess* f. *A good first guess is the "fully rough" value (wholly turbulent) for* $\varepsilon/d = 0.0002$ *from Moody chart. It is* $f \approx 0.015$. *Now from Darcy-Weisbach formula (4) we have*

$$h_f = f\frac{L}{D}\frac{v^2}{2g} \quad \Rightarrow \quad fv^2 = 0.471$$

$$f \approx 0.015 \qquad v = \sqrt{0.471/0.014} = 5.8\ m/s \qquad Re = vd/\nu \approx 87000$$
$$f_{new}(87000) = 0.0195 \quad v = \sqrt{0.471/0.0195} = 4.91\ m/s \qquad Re = vd/\nu \approx 73700$$
$$f_{new}(73700) = 0.0201 \quad v = \sqrt{0.471/0.0201} = 4.84\ m/s \qquad Re = vd/\nu \approx 72600$$

$$f(72600) = 0.0201, \text{ so we can accept } v = 4,84\ m/s, \quad q_v = v\left(\frac{\pi d^2}{4}\right) = 0,342\ m^3/s.$$

1.1.2. Calculating head losses in non-circular pipes

The basic approach to all piping systems is to write the Bernoulli equation between two points, connected by a streamline, where the conditions are known. For example, between the surface of a reservoir and a pipe outlet.

$$h_0 + \frac{P_0}{\rho g} + \frac{V_0^2}{2g} + \Delta h_p = h_1 + \frac{P_1}{\rho g} + \frac{V_1^2}{2g} + \Delta h_f + \Delta h_m$$

The total head at point 0 must match with the total head at point 1, adjusted for any increase in head due to pumps, losses due to pipe friction and so-called "minor losses" due to entries, exits, fittings, etc. Pump head developed is generally a function of the flow through the system, with head rise decreasing with increasing flow through the pump.

Friction Losses in Pipes

Friction losses are a complex function of the system geometry, the fluid properties and the flow rate in the system. By observation, the head loss is roughly proportional to the square of the flow rate in most engineering flows (fully developed, turbulent pipe flow). This observation leads to the Darcy-Weisbach equation for head loss due to friction:

$$\Delta h_f = f \frac{L}{D} \frac{V^2}{2g}$$

which defines the friction factor, f. f is insensitive to moderate changes in the flow and is constant for fully turbulent flow. Thus, it is often useful to estimate the relationship as the head being directly proportional to the square of the flow rate to simplify calculations.

$$f = f\left(\mathrm{Re}, \frac{\varepsilon}{D}, \text{pipe cross-section}\right) \qquad \mathrm{Re} = \frac{VD}{\nu}$$

Reynolds Number is the fundamental dimensionless group in viscous flow. Velocity times Length Scale divided by Kinematic Viscosity.

Relative Roughness relates the height of a typical roughness element to the scale of the flow, represented by the pipe diameter, D.

Pipe Cross-section is important, as deviations from circular cross-section will cause secondary flows that increase the pressure drop. Non-circular pipes and ducts are generally treated by using the hydraulic diameter,

$$D_H \equiv \frac{4A}{P} = \frac{4 \text{ x the cross - sectional area}}{\text{the wetted perimeter of the pipe}}$$

in place of the diameter and treating the pipe as if it were round.

For laminar flow, the head loss is proportional to velocity rather than velocity squared, thus the friction factor is inversely proportional to velocity.

$$\text{Circular Pipes: } f = \frac{64}{\text{Re}}$$

$$\text{Non - Circular Pipes: } f = \frac{k}{\text{Re}},\ 48 \le k \le 96$$

	Geometry Factor k
Square	56.91
2:1 Rectangle	62.19
5:1 Rectangle	76.28
Parallel Plates	96.00

The Reynolds number must be based on the hydraulic diameter. Blevins (Applied Fluid Dynamics Handbook, table 6-2, pp. 43-48) gives values of k for various shapes. For turbulent flow, Colebrook (1939) found an implicit correlation for the friction factor in round pipes. This correlation converges well in few iterations. Convergence can be optimized by slight under-relaxation.

$$\frac{1}{\sqrt{f}} = -0.869 \ln\left(\frac{\varepsilon / \text{D}}{3.7} + \frac{2.523}{\text{Re}\sqrt{f}}\right)$$

The familiar Moody Diagram is a log-log plot of the Colebrook correlation on axes of friction factor and Reynolds number, combined with the f=64/Re result from laminar flow.

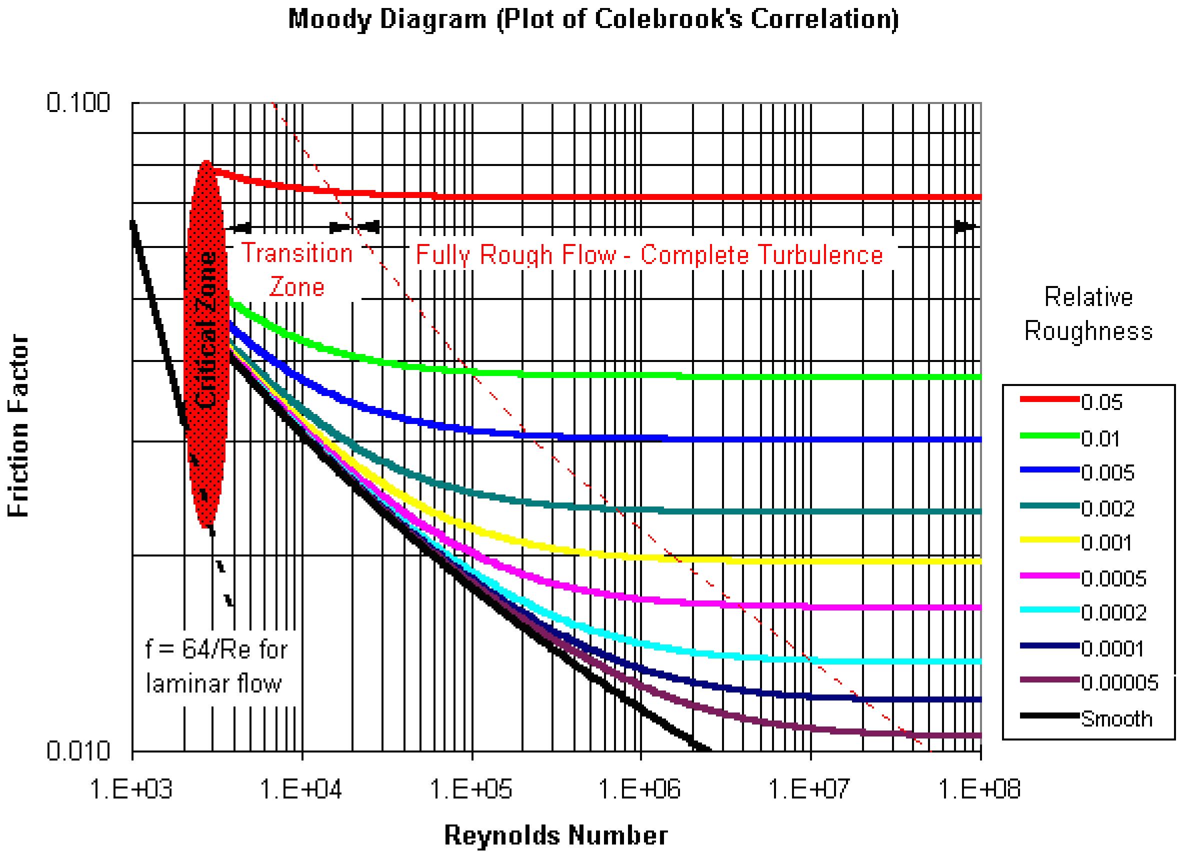


Figure1. 2 moody diagram

An explicit approximation

$$f = \frac{1.325}{\left(\ln\left(\frac{\varepsilon / D}{3.7} + \frac{5.74}{Re^{0.9}}\right)\right)^2} \qquad 10^{-6} \le \varepsilon / D \le 10^{-2} \qquad 5000 \le Re \le 10^8$$

provides values within one percent of Colebrook over most of the useful range.

Calculating Head Loss for a Known Flow

From Q and piping determine Reynolds Number, relative roughness and thus the friction factor. Substitute into the Darcy-Weisbach equation to obtain head loss for the given flow. Substitute into the Bernoulli equation to find the necessary elevation or pump head.

Calculating Flow for a Known Head

Obtain the allowable head loss from the Bernoulli equation, then start by guessing a friction factor. (0.02 is a good guess if you have nothing better.) Calculate the velocity from the Darcy-Weisbach equation. From this velocity and the piping characteristics, calculate Reynolds Number, relative roughness and thus friction factor. Repeat the calculation with the new friction factor until sufficient convergence is obtained. Q = VA.

Minor Losses

Although they often account for a major portion of the head loss, especially in process piping, the additional losses due to entries and exits, fittings and valves are traditionally referred to as minor losses. These losses represent additional energy dissipation in the flow, usually caused by secondary flows induced by curvature or recirculation. The minor losses are any head loss present *in addition to the head loss for the same length of straight pipe.*

Like pipe friction, these losses are roughly proportional to the square of the flow rate. Defining K, the loss coefficient, by

$$\Delta h_m = \Sigma K \frac{V^2}{2g}$$

allows for easy integration of minor losses into the Darcy-Weisbach equation. K is the sum of all of the loss coefficients in the length of pipe, each contributing to the overall head loss.

Although K appears to be a constant coefficient, it varies with different flow conditions. Factors affecting the value of K include:

- the exact geometry of the component in question
- the flow Reynolds Number
- proximity to other fittings, etc. (Tabulated values of K are for components in isolation - with long straight runs of pipe upstream and downstream.)

Some very basic information on K values for different fittings is included with these notes and in most introductory fluid mechanics texts. For more detail see e.g. Blevins, pp. 55-88.

To calculate losses in piping systems with both pipe friction and minor losses use

$$\Delta h_f = \left(f \frac{L}{D} + \Sigma K \right) \frac{V^2}{2g}$$

in place of the Darcy-Weisbach equation. The procedures are the same except that the K values may also change as iteration progresses.

1.1.3. Calculating minor energy losses associated with enlargements, contraction, valves, fittings and bends

Two types of energy loss predominate in fluid flow through a pipe network; major losses, and minor losses. Major losses are associated with frictional energy loss that is caused by the viscous effects of the medium and roughness of the pipe wall. Minor losses, on the other hand, are due to pipe fittings, changes in the flow direction, and changes in the flow area. Due to the complexity of the piping system and the number of fittings that are used, the head loss coefficient (K) is empirically derived as a quick means of calculating the minor head losses.

The term "minor losses", used in many textbooks for head loss across fittings, can be misleading since these losses can be a large fraction of the total loss in a pipe system. In fact, in a pipe system with many fittings and valves, the minor losses can be greater than the major (friction) losses. Thus, an accurate K value for all fittings and valves in a pipe system is necessary to predict the actual head loss across the pipe system. K values assist engineers in totaling all of the minor losses by multiplying the sum of the K values by the velocity head to quickly determine the total head loss due to all fittings. Knowing the K value for each fitting enables engineers to use the proper fitting when designing an efficient piping system that can minimize the head loss and maximize the flow rate. Loss coefficient (K) for a range of pipe fittings, including several bends, a contraction, an enlargement, and a gate valve

The resistance to flow in a pipe network causes loss in the pressure head along the flow. The overall head loss across a pipe network consists of the following:

- Major losses (hmajor), and
- Minor losses (hminor)

(i) Major losses Major losses refer to the losses in pressure head of the flow due to friction effects. Such losses can be evaluated by using the Darcy-Weisbach equation:

$$h_{major} = f\frac{Lv^2}{2gD} \qquad (1)$$

where f isthe Darcy friction factor, L is the length of the pipe segment, v is the flow velocity, D is the diameter of the pipe segment, and g is acceleration due to gravity. Equation (1) is valid for any fully-developed, steady and incompressible flow.

The friction factor fcan be calculated by the following empirical formula, known as the Blasius formula, valid for turbulent flow in smooth pipes with $Re_D < 10^5$:

$$f = 0.316(Re_D)^{-0.25} \qquad (2)$$

(ii) Minor losses

In a pipe network, the presence of pipe fittings such as bends, elbows, valves, sudden expansion or contraction causes localized loss in pressure head. Such losses are termed as minor losses. Minor losses are expressed using the following equation:

$$h_{minor} = K\frac{v^2}{2g} \qquad (3)$$

where Kis called the Loss Coefficient of the pipe fitting under consideration. Minor losses are also expressed in terms of the equivalent length of a straight pipe (L_{eq}) that would cause the same head loss as the fitting under consideration:

$$h_{minor} = K\frac{v^2}{2g} = f\frac{L_{eq}v^2}{2gD}$$

$$L_{eq} = K\frac{D}{f} \qquad (4)$$

or

We shall determine the head losses across sudden enlargement, sudden contraction, sharp bend (90 ° elbow), smooth bend, and a straight section.

Loss of head due to sudden enlargement: This is the energy loss due to sudden enlargement. Sudden enlargement in the diameter of pipe results in the formation of eddies in the flow at the corners of the enlarged pipe(Fig.1).This results in the loss of head across the fitting

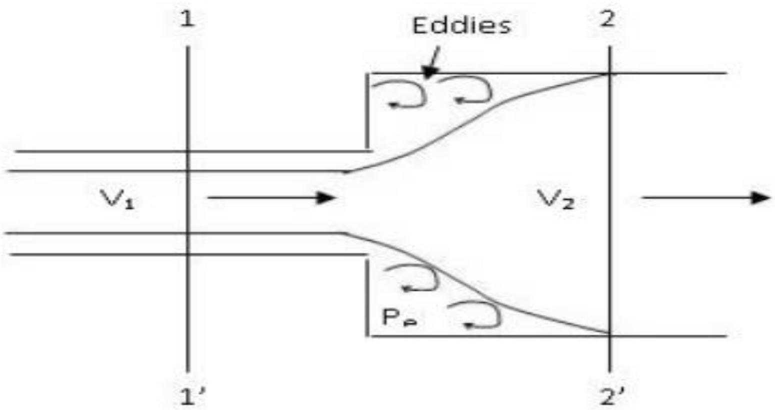


Figure 1.3 Sudden Expansions

Loss of head due to sudden contraction: This is the energy loss due to sudden contraction. In reality, the head loss does not take place due to the sudden contraction but due to the sudden enlargement, which takes place just after vena-contracta (Fig. 2).

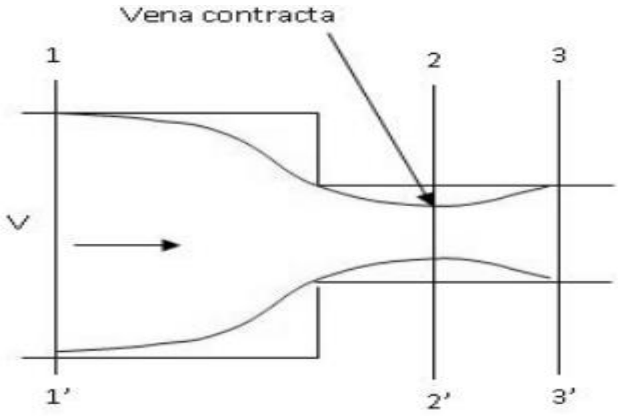


Figure 1.4 Sudden Contraction

Loss of head due to bend in pipe:

This is the energy loss due to bend. When a bend is provided in the pipeline, there is a change in direction of the velocity of flow (figures 1.5 and 1.6). Due to this, the flow separates from the walls of the bend and formation of eddies takes place.

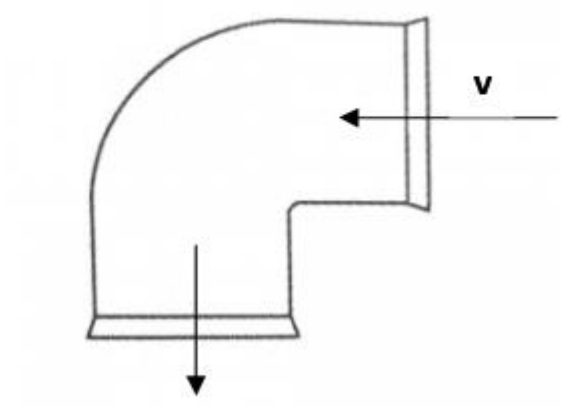


Figure 1.5 Sharp Bend (90° elbow)

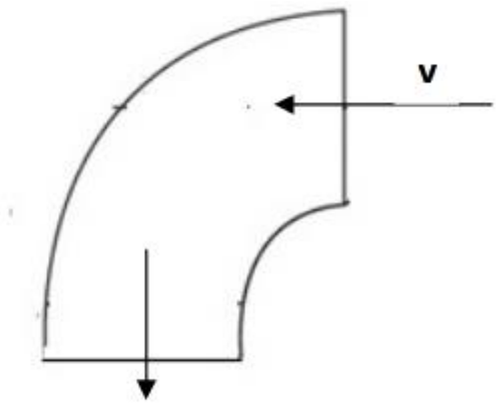


Figure 1.6 Smooth Bend

Figure 1.7 shows the schematic layout of the pipe network to be used in the present study

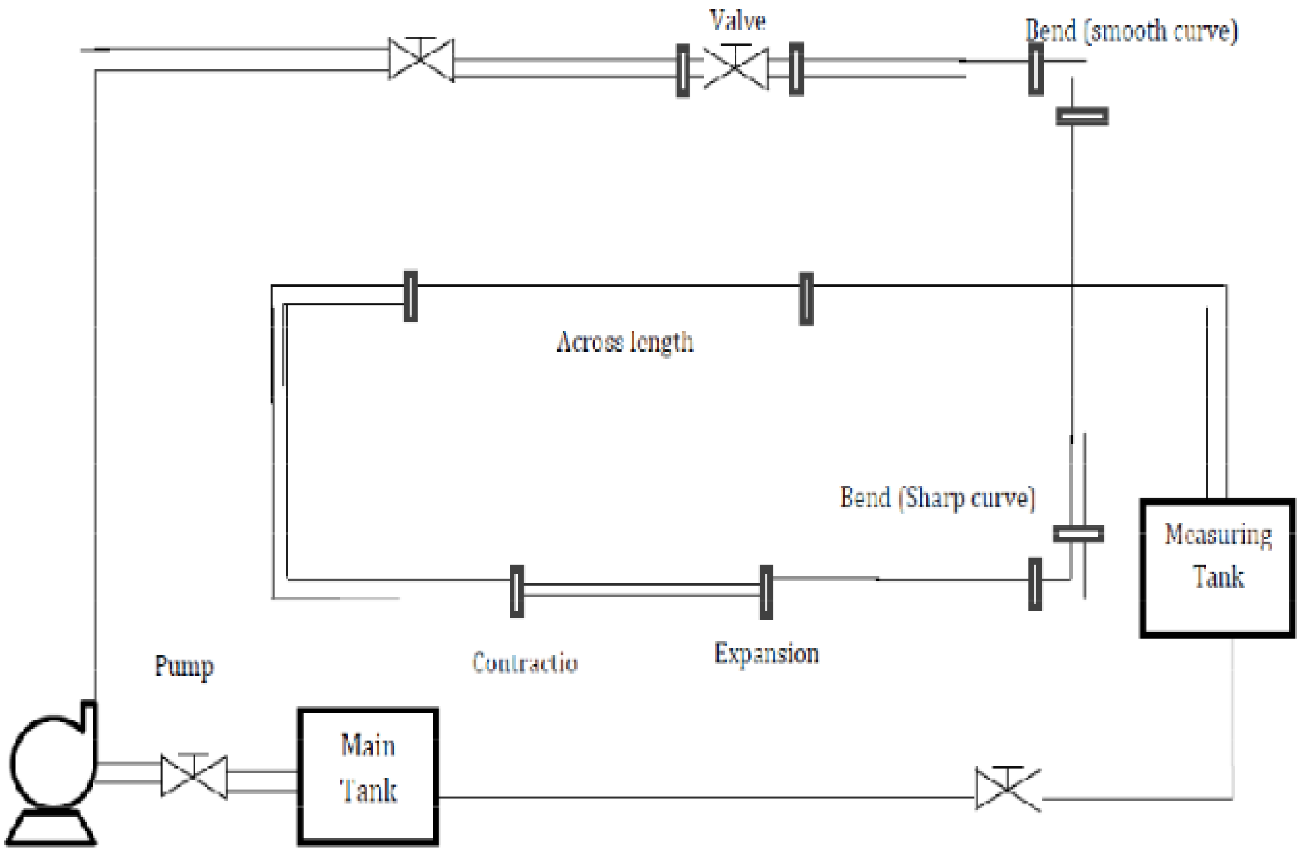
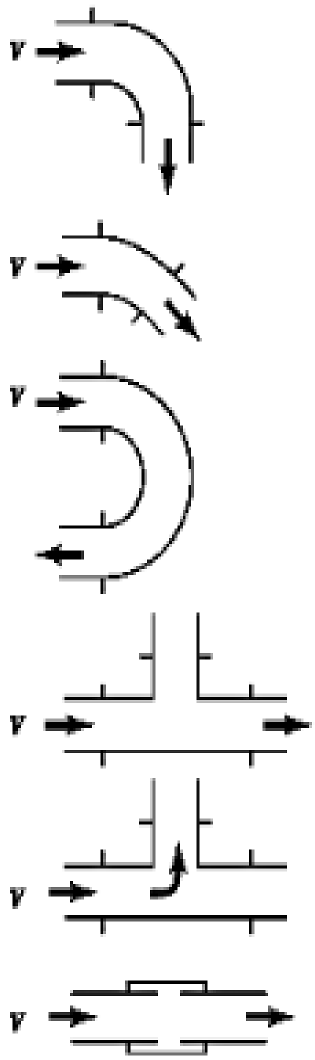


Figure 1,7 Schematic layout of pipe network with fittings

Table 1.1 loss coefficients for pipe components

Loss Coefficients for Pipe Components $\left(h_L = K_L \frac{V^2}{2g}\right)$

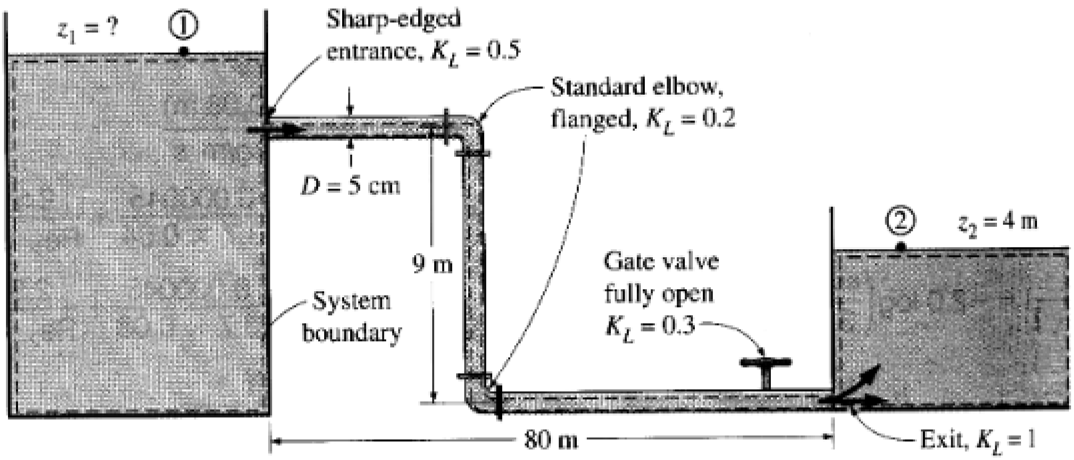
Component	K_L
a. Elbows	
Regular 90°, flanged	0.3
Regular 90°, threaded	1.5
Long radius 90°, flanged	0.2
Long radius 90°, threaded	0.7
Long radius 45°, flanged	0.2
Regular 45°, threaded	0.4
b. 180° return bends	
180° return bend, flanged	0.2
180° return bend, threaded	1.5
c. Tees	
Line flow, flanged	0.2
Line flow, threaded	0.9
Branch flow, flanged	1.0
Branch flow, threaded	2.0
d. Union, threaded	0.08
***e. Valves**	
Globe, fully open	10
Angle, fully open	2
Gate, fully open	0.15
Gate, $\frac{1}{4}$ closed	0.26
Gate, $\frac{1}{2}$ closed	2.1
Gate, $\frac{3}{4}$ closed	17
Swing check, forward flow	2
Swing check, backward flow	∞
Ball valve, fully open	0.05
Ball valve, $\frac{1}{3}$ closed	5.5
Ball valve, $\frac{2}{3}$ closed	210



Self-Check -1	Written Test

Direction: write the answer for the following questions. Use the Answer sheet provided

1. Determining the pipe diameter when the pipe length and flow rate are given for a specified pressure drop. Oil, with ρ=950kg/m3and6==2E−5m2/s, flows through a30−cm-diameterpipe 100 m long with a head loss of 8 m. The roughness ratio is ε/d=0.0002. Find the average velocity and flow rate.
2. Water at10oC flow from a large reservoir to a small one through a 5-cm-diameter cast iron piping system shown in figure . Determine the elevationz1for the flow rate of6l/s.



3. Use the Flow Master program to compare the head loss computed by the Hazen-Williams equation to the head loss computed by the Darcy-Weisbach equation for a pressure pipe having the following characteristics: 12-in diameter cast iron pipe (new) one mile inlength with a flow rate of 1,200 gallons per minute (with water at 65°F).

Answer Sheet-1

Name: _________________Date: ______________

Score = ___________

Rating: ____________

Answer

1. ____
____
____
____
____
____
____
2. ____
____
____
____
____
____
____
____
3. ____
____
____
____
____
____
___

Water and Measurement

Hydraulic problems concerning fluid flow are generally handled by accounting in terms of energy per pound of flowing water. Energy measured in this form has units of feet of water. The total amount of energy is that caused by motion, or velocity head, $V^2/2g$, which has units of feet, plus the potential energy head, Z, in feet, caused by elevation referenced to an arbitrary datum selected as reference zero elevation, plus the pressure energy head, h, in feet. The head, h, is depth of flow for the open channel flow case and p/γ defined by equation 2-2 for the closed conduit case. This summation of energy is shown for three cases on figure 2-1.

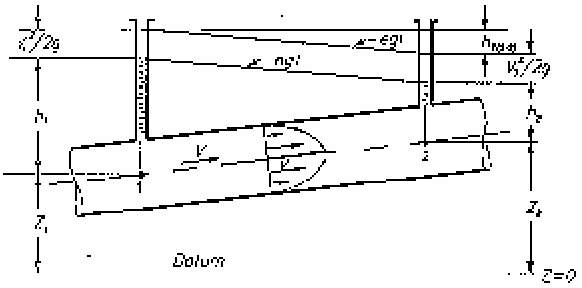


Figure 2-1a -- Energy balance in pipe flow.

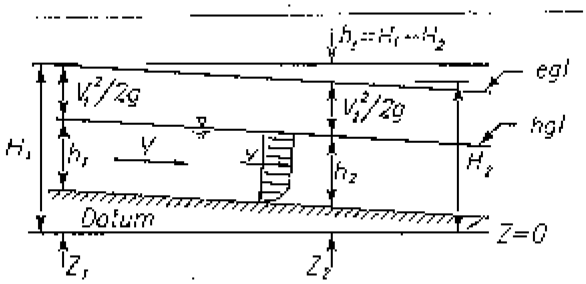


Figure 2-1b -- Energy balance in open channel flow.

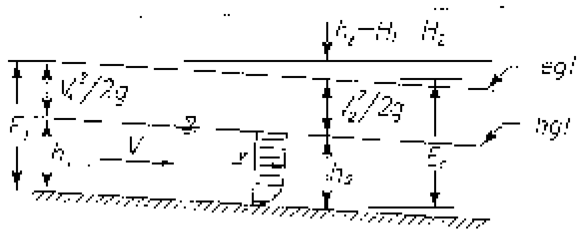


Figure 2-1c -- Specific energy balance.

Figures 2-1a and 2-1b show the total energy head, H_1; for example, at point 1, in a pipe and an open channel, which can be written as:

$$H_1 = h_1 + \frac{V_1^2}{2g} + Z_1 \qquad (2\text{-}1)$$

At another downstream location, point 2:

$$H_2 = h_2 + \frac{V_2^{\,2}}{2g} + Z_2 \qquad (2\text{-}2)$$

Energy has been lost because of friction between points 1 and 2, so the downstream point 2 has less energy than point 1. The energy balance is retained by adding a head loss, $h_{f(1\text{-}2)}$. The total energy balance is written as:

$$h_1 + \frac{V_1^{\,2}}{2g} + Z_1 = h_2 + \frac{V_2^{\,2}}{2g} + Z_2 + h_{f\,(1\text{-}2)} \qquad (2\text{-}3)$$

The upper sloping line drawn between the total head elevations is the energy grade line, *eg l*. The next lower sloping solid line for both the pipe and open channel cases shown on figure 2-3 is the hydraulic grade line, *hgl*, which is also the water surface for open channel flow, or the height to which water would rise in piezometer taps for pipe flow.

A special energy form is commonly used in hydraulics in which the channel invert is selected as the reference Z elevation (figure 2-3c). Thus, Z drops out, and energy is the sum of depth, h, and velocity head only. Energy above the invert expressed this way is called specific energy, E. This simplified form of energy equation is written as:

$$\text{Specific energy} = E = \frac{V^2}{2g} + h \qquad (2\text{-}4)$$

Equations 2-3 and 2-4 lead to several interesting conclusions. In a fairly short pipe that has little or insignificant friction loss, total energy at one point is essentially equal to the total energy at another point. If the size of the pipeline decreases from the first point to the second, the velocity of flow must increase from the first point to the second. This increase occurs because with steady flow, the quantity of flow passing any point in the completely filled pipeline remains the same. From the continuity equation (equation 2-4), when the flow area decreases, the flow velocity must increase.

The second interesting point is that when the velocity increases in the smaller section of the pipeline, the pressure head, h, decreases. At first, this decrease may seem strange, but equation 2-3 shows that when $V^2/2g$ increases, h must decrease proportionately because the total energy from one point to another in the system remains constant, neglecting friction loss. The fact that the pressure does decrease when the velocity in a given system increases is the basis for tube-type flow measuring devices.

In open channel flow where the flow accelerates, more of its supply of energy becomes velocity head, and depth must decrease. On the other hand, when the flow slows down, the depth must increase.

An example of accelerating flow with corresponding decreasing depth is found at the approach to weirs. The drop in the water surface is called drawdown. Another example occurs at the entrance to inverted siphons or conduits where the flow accelerates as it passes from the canal, through a contracting transition, and into the siphon barrel. An example of decelerating flow with a rising water surface is found at the outlet of an inverted siphon, where the water loses velocity as it expands in a transition back into canal flow.

Flumes are excellent examples of measuring devices that take advantage of the fact that changes in depth occur with changes in velocity. When water enters a flume, it accelerates in a converging section. The acceleration of the flow causes the water surface to drop a significant amount. This change in depth is directly related to the rate of flow

Self check 2	Written test.

Direction : write the answer for the following questions. Use the Answer sheet provided

1. What are excellent examples of measuring devices that take advantage of the fact that changes in depth occur with changes in velocity?
2. The drop in the water surface is called ________________.
3. Energy above the invert expressed this way is called ______________ ,it simplified with what?

Note: **Satisfactory rating - 5 points** **Unsatisfactory - below 5 points**

You can ask you teacher for the copy of the correct answers.

Answer Sheet-1

Name: _________________Date: ______________

Score = ___________

Rating: ____________

Answer

1. ____
____
____
2. ____
____
3. ____
____

Information sheet 3	Applying Hydraulic software

Computer Applications

It is very important for students (and practicing engineers) to fully understand the methodologies behind hydraulic computations. Once these concepts are understood, the solution process can become repetitive and tedious—the type of procedure that is well-suited to computer analysis. There are several advantages to using computerized solutions for common hydraulic problems: The amount of time to perform an analysis can be greatly reduced. Computer solutions can be more detailed than hand calculations. Performing a solution manually often requires many simplifying assumptions. The solution process may be less error-prone. Unit conversion and the rewriting of equations to solve for any variable are just two examples of mistakes frequently introduced with hand calculations. A well-tested computer program helps to avoid these algebraic and numeric errors. The solution is easily documented and reproducible. Because of the speed and accuracy of a computer model, more comparisons and design trials can be performed. The result is the exploration of more design options, which eventually leads to better, more efficient designs. In order to prevent an "overload" of data, this chapter deals primarily with steady-state computations. After all, an introduction to hydraulic calculations is tricky enough without throwing in the added complexity of a constantly changing system

The assumption that a system is under steady-state conditions is oftent perfectly acceptable. Minor changes that occur over time or irregularities in a channel cross-sectionare frequently negligible, and a more detailed analysis may not be the most efficient or effective use of time and resources. There are circumstances when an engineer may be called upon to provide a more detailed analysis, including unsteady flow computations. For a storm sewer, the flows may rise and fall over time as a storm builds and subsides. For water distribution piping, a pressure wave may travel through the system when a valve is closed abruptly (the same "water-hammer" effect can probably be heard in your house if you close a faucet quickly). As an engineer, it is important to understand the purpose of an analysis; otherwise, appropriate methods and tools to meet that purpose cannot be selected.

Flow Master

Flow Master is an easy-to-use program that helps civil engineers with the hydraulic design and analysis of pipes, gutters, inlets, ditches, open channels, weirs, and orifices. Flow Master computes flows and pressures in conduits and channels using common head loss equations such as Darcy-We is bach, Manning's, Kutter's, and Hazen-Williams. The program's flexibility allows the user to choose an unknown variable and automatically compute the solution after entering known parameters. Flow Master also calculates rating tables and plots curves and cross-sections. You can view the output on the screen, copy it to the Windows clipboard, save it to a file, or print it on any standard printer. Flow Master data can also be viewed and edited using tabular reports called Flex Tables. Flow Master enables you to create an unlimited number of worksheets to analyze uniform pressure-pipe or open-channel sections, including irregular sections (such as natural streams or odd-shaped man-made sections). Flow Master does not work with networked systems such as a storm sewer network or a pressure pipe network. For these types of analyses, Storm CAD, Water CAD, or Sewer CAD should be used instead. The theory and background used by Flow Master have been reviewed in this chapter and can be accessed via the Flow Master on-line help system. General information about installing and running Haestad Methods software can be found in Appendix A. Flow Master replaces solutions such as nomographs, spreadsheets, and BASIC programs. Because Flow Master gives you immediate results, you can quickly generate output for a large number of situations. For example, you can use Flow Master to: Analyze various hydraulic designs. Evaluate different kinds of flow elements Generate professional-looking reports for clients and review agencies Computer Applications in Hydraulic Engineering

Tutorial Example

The following solution gives step-by-step instructions on how to solve an example problem using the Flow Master computer program (included on the CD-ROM that accompanies this textbook) developed by Haestad Methods. Problem Statement Using Manning's equation, design a triangular concrete channel with equal side slopes, a longitudinal slope of 5%, a peak flow capacity of 0.6 m3/s, and a maximum depth of0.3. Also, design a concrete trapezoidal channel with equal side slopes and a base width of 0.2 that meets the same criteria. Create a cross-section of each channel and a curve of discharge versus depth for each channel. Assume the water is at 20°C. Solution Upon opening Flow Master, click Create New Project in the Welcome to Flow Master dialog. Enter a filename and click Save. $ffffffff$

Select Triangular Channel from the Create a New Worksheet dialog and click OK. In the Triangular Channel dialog, select Manning's Formulafrom the FrictionMethod pull-down menu. Enter a label for the worksheet and clickOK.Select Global Options from the Options menu and change the unit system to System International, if it has not already been done, by selecting it from the pull-down menu in the Unit System field. Click OK to exit the dialog. If you changed the unit system, you will be prompted to confirm the unit change. Click Yes. The worksheet dialog should appear. Because discharge, channel slope, and depth aregiven, the variable you need to solve for is the side slopes of the channel. Select Equal Side Slopes from the Solve for: menu at the top of the dialog. Enter the channel slope (you can change the units to percent by double left-clicking the units), depth, and discharge into the appropriate fields and select the Manning's n for concrete. ClickSolve. The equal side slopes should be 1.54 H:V. To design the trapezoidal section, first click Close on the triangular section worksheet to save it. Then click Create... at the bottom of the Worksheet List. Select the Trapezoidal Channel and OK. Repeat the same steps as before to design the triangular channel. The equal side slopes should be 0.80 H:V. Click Close to exit the worksheet.

Creating Channel Cross-Sections

- Open the triangular section worksheet by highlighting it and selecting the Open button *ƒ*
- Click the Report button on the bottom of the triangular channel worksheet. Select Cross Section..
- Type a report title and click OK. Figure 3-1provides a graphical representation of both the triangular and trapezoidal channel designs. Click Print if you want to print a copy of the cross-section. Click Close to exit the report. Finally, click Close to exit the triangular section worksheet. The drawing for the trapezoidal section cross-section is created in the same way from its worksheet.

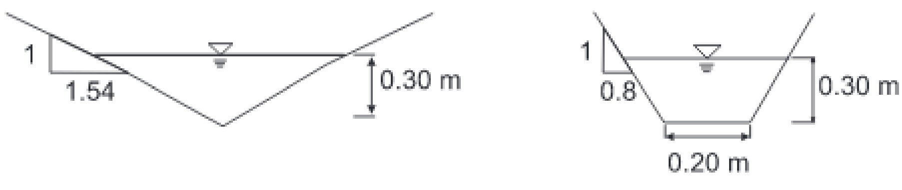


Figure 3-1: Triangular and Trapezoidal Channel Designs

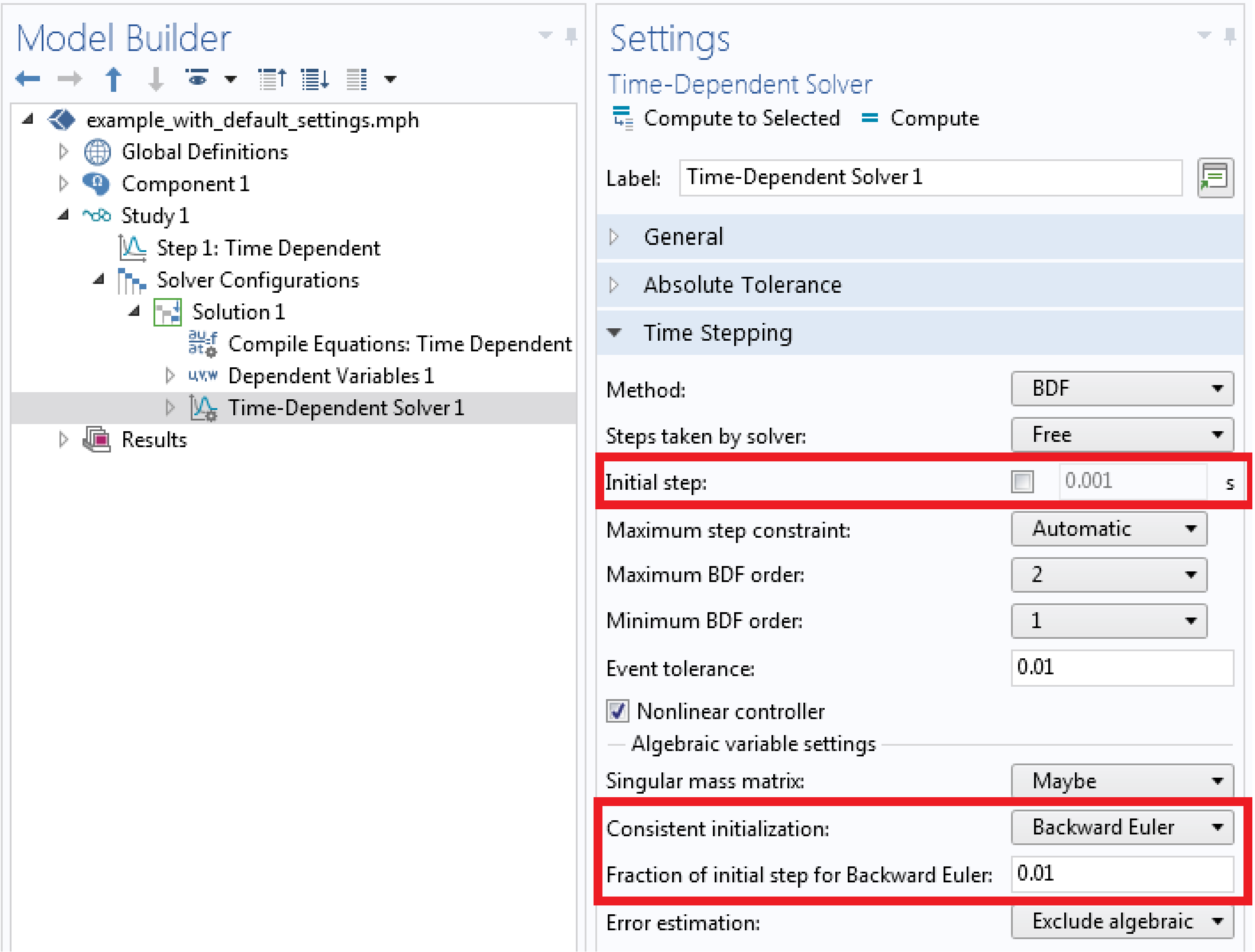
Creating Discharge versus Depth Curves

- Open the trapezoidal section worksheet by highlighting it and selecting the Open button.
- Because discharge needs to be on the y-axis (the ordinate) of the graph, you need to change Equal Side Slopes in the Solve For: field to Discharge.
- Click Report... at the bottom of the worksheet. Select Rating Curve from the drop-down list.
- In the Graph Setup dialog, select Discharge from the pull-down menu in the field labeled Plot. Select Depth from the pull-down menu in the field labeled vs.
- To scale the plot properly, make the minimum depth 0 m and the maximum depth 0.3 m. Choose an increment based on how smooth you want to make the curve. An increment of 0.1 will give the plot 3 points, an increment of 0.01 will give the plot 30 points, etc. Click

OK. Click Print Preview at the top of the window, and then click Print to print out a report featuring the rating curve you have just created. Click Close to exit the Print Preview window. Click Close again to exit the Plot Window.

Model software

This is undesirable as it leads to relatively long solution times.



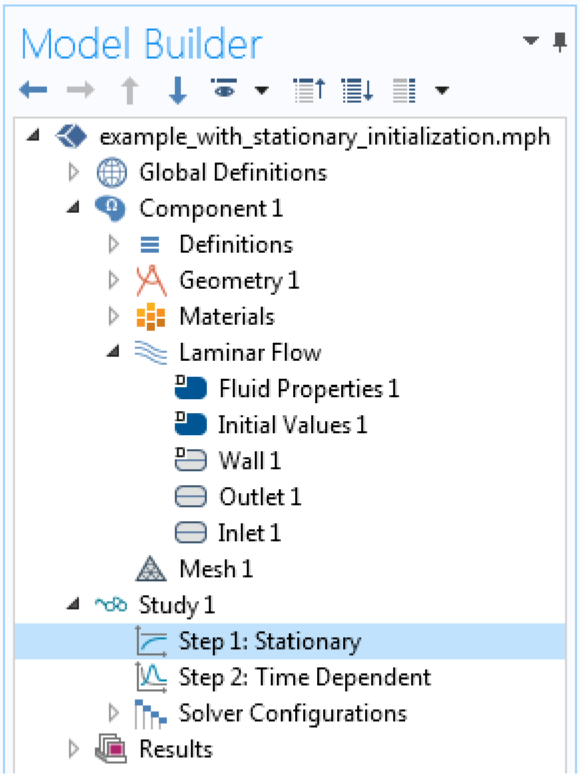
The default settings for the initial timestep size and consistent initialization.

In many cases this consistent initialization is not strictly needed and it might be better to turn it off and to ensure that your model smoothly ramps all loads and boundary conditions from consistent initial values. This Knowledgebase presents two ways to do so.

Note, however, that if your boundary conditions truly are inconsistent with each other and the physics, then the resolutions in this Knowledgebase will not apply. Always check carefully the applied loads and boundary conditions for consistency. The recommendations in this Knowledgebase apply to well-posed problems.

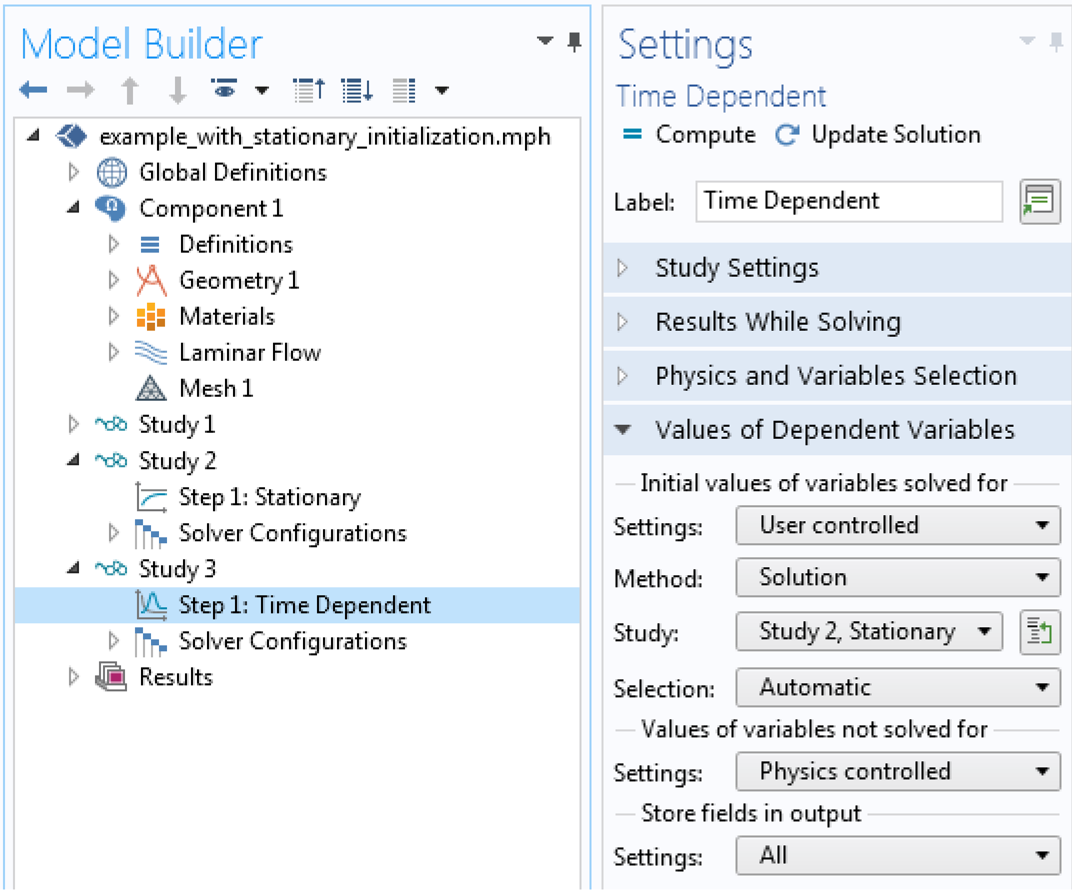
Method 1: Initialize the time dependent study with a stationary study

A single **Study** can include multiple **Steps**, and by default the results of each step are passed on to the following step as initial values. Adding a stationary step before a transient study step will thus solve first for the fields under the steady-state assumption, and these provide a consistent initial value for the transient step, overriding the initial values specified within the **Initial Values** feature in the physics interface. As long as the two steps are within the same study, as shown in the screenshot below, no other settings changes are needed. When the study is solved, both steps will be re-computed.



Using a stationary step step to compute initial values for the transient study step.

It is also possible to split the stationary step and the transient step into two different studies. This requires that the **Values of Dependent Variables** in the transient study are manually set to refer to the results of the stationary study, as shown in the screenshot below. The corresponding model file can be downloaded here.



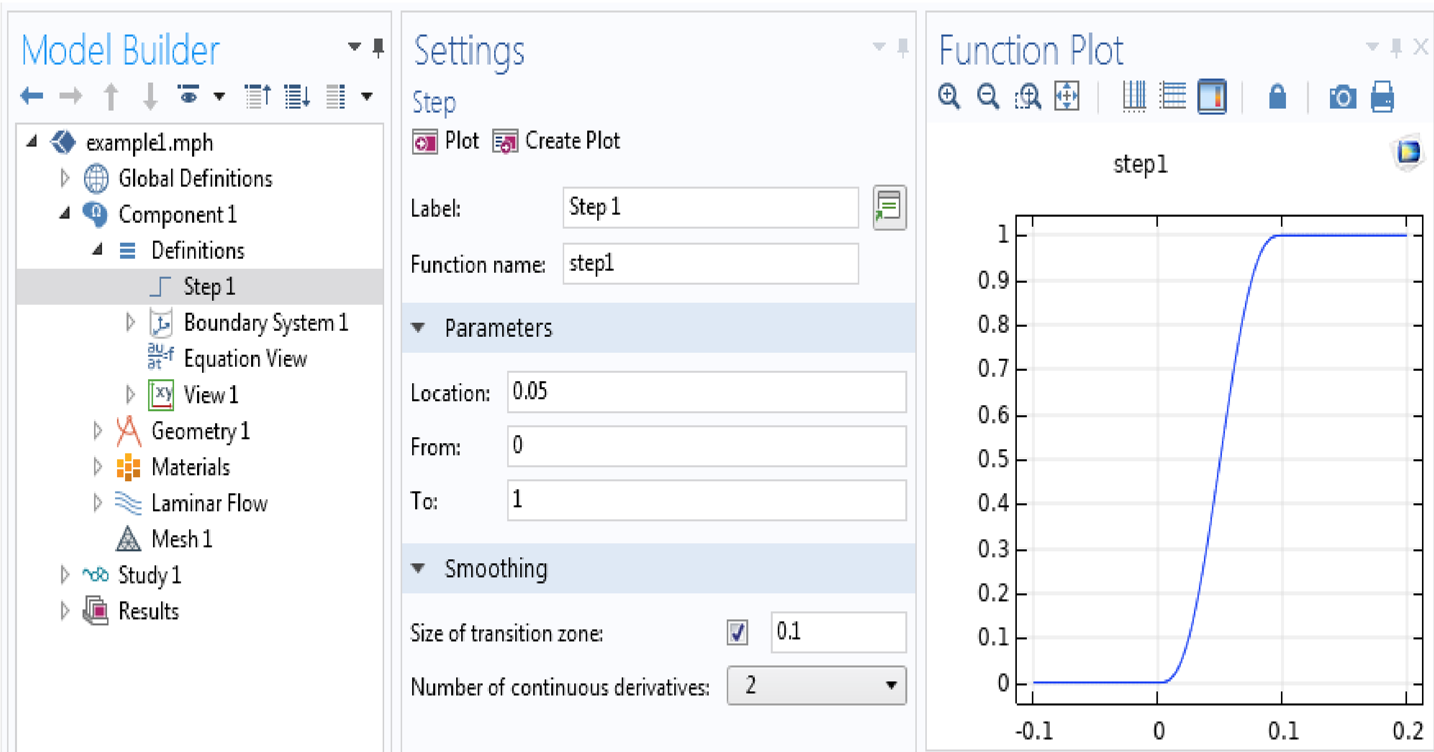
Adjusting the settings of a time dependent study to use the initial values computed from a stationary study.

This strategy has a few drawbacks, however. First, a stationary solution might not exist at all. This is particularly common for fluid flow models near the onset of turbulence, where steady boundary conditions result in an unsteady (time-varying) flow field. It can also be that a steady state solution is quite numerically difficult to find, see Knowledgebase 103: Improving Convergence of Nonlinear Stationary Models for how to address such cases. Second, the objective of the transient model may be to investigate the startup behavior, as the system evolves from the at-rest state. In either case, use the second method, described below.

Method 2: Ramp up the boundary conditions over time

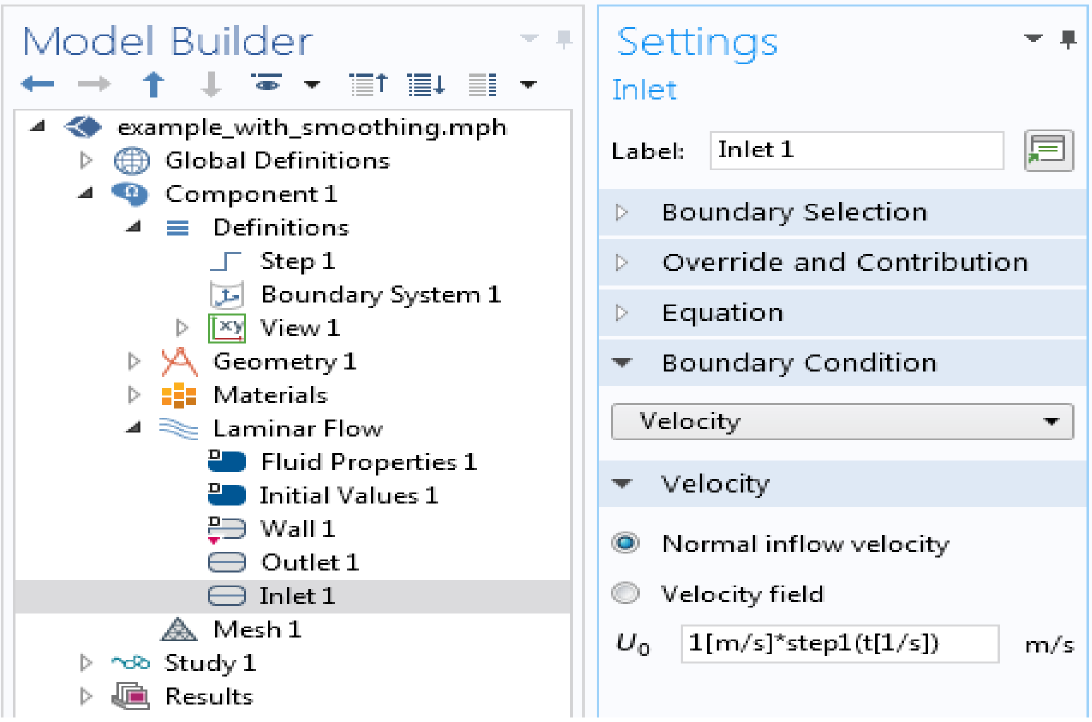
The loads and boundary conditions on a transient model can be ramped up from values that are consistent with the initial values. The most common case would be an at-equilibrium system, where the initial values are zero everywhere. Use the built-in **Step** function with smoothing, as shown in the screenshot below. There several other built-in functions that also include a **Smoothing** option, as described in Knowledgebase 905: Modeling of step transitions.

These all, by default, will have a time-derivative equal to zero at the start of the smoothing zone, which is helpful.



The settings of the Step function with smoothing.

Use this smoothed step function to modify the loads and boundary conditions, as shown in the screenshot below. The corresponding model file can be downloaded [here](#).



The smoothed step function is used to ramp the boundary condition.

You will need to choose a timespan for the smoothing that is physically realistic for the the problem at hand. For example, in the case of laminar fluid flow, you should not ramp the velocity so quickly as to introduce a supersonic shock. Or, for an electromagnetic wave problem, do not introduce a ramping that is faster than the speed of light. See also Knowledgebase 1244: Solving Wave-Type Problems with Step Changes in the Loads. It can also instead be reasonable to introduce a very gradual ramping, depending upon your desired model outputs.

Note that not all problems will require this type of smoothing. Some problems, particularly heat transfer problems that do not involve convection, can be solved with step changes in the loads. If these step changes happen during the simulation time-span you should use **Events** to accurately model these cases, as described in Knowledgebase 1245: Solving Models with Pulsed Loads in Time.

General Remarks

Once either of the above techniques is implemented, the issue of consistent initialization should be resolved, and you should not have to alter the study settings.

If you do continue to have issues with convergence, it can be that your model is not meshed finely enough, in which case you should perform a mesh refinement study as described in Knowledgebase 1261: Performing a Mesh Refinement Study. It can also be that your model is highly nonlinear, in which case see Knowledgebase 1127: Improving convergence in nonlinear time dependent models.

Self-Check -3	Written Test

Direction I: Choose the best answer for the following questions. Use the Answer sheet provided in the next page:Each question worth one point

1. Which step is considered developing sampling plan?

A/ Selecting sampling size B/Identifying of parameters

C/ Design of sampling scheme D/ All

2. Which one of these is **not** a disadvantage of composite samples?

A/ Loss of analyze relationships in individual samples

B/ Reduced costs of analyzing of samples

C/potential dilution of analyzes below detection levels

D/ Increase the possibility of analyze interaction.

3-------------- is atype of sample usually taken when you want information specific to a particular sampling location, time or distinct areas within a sampling location:

A/ Composite sample B/ Grab sample C/ Analyze D/ All

4. Types of samples usually taken when we want an average representation of a sampling location or time are-------. A/Grab sample B/ Composite sample C/Discreet grab sample D/ None

5. A properly taken grab sample is a snap shot of the quality of the water at the exact time and place the sample was taken.

A/ True B/ False

Note: **Satisfactory rating - 5 points** **Unsatisfactory - below 5points**

Answer Sheet-1

Name: ___________________________ Date: ________________

Choice Questions

1________ 4.___________

2.________ 5.___________

3________

Score = ___________

Information 4	Identifying standard process inconsistent data on flow conditions.

Introduction

A common mistake when setting up transient models is to have initial conditions that are inconsistent with the loads and boundary conditions. This occurs most often when running transient fluid flow studies, but the same type of issue can occur in any time-dependent model.

You may observe the solver taking very small time steps at the beginning of the simulation, or the solver will report an error message similar to:

```
Failed to find consistent initial values.
Last time step is not converged.
```

Solution

Background

To illustrate this issue and its resolutions, consider a simple fluid flow model as diagrammed below. Solve for **Laminar Flow** of a fluid ( ) in a two-dimensional channel (dimensions 3m x 1m) with an inlet on the left and an outlet on the right. Specify a velocity of 1 m/s at the inlet and a zero-pressure boundary condition at the outlet. The initial values in the channel are left to their default values, i.e., (u,v)=(0,0) and p=0.

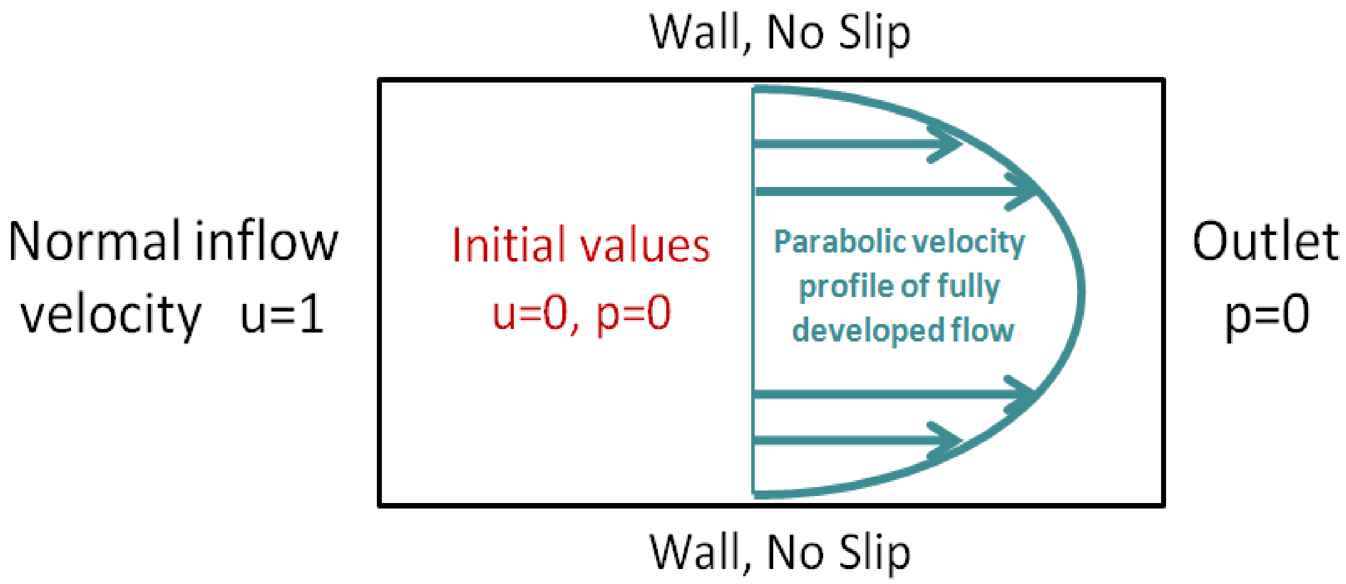


Figure 4.1 parabolic velocity profile of fully developed flow

These boundary conditions lead to a mismatch between the value of the velocity at the inlet u=1m/s and the value of the velocity inside the channel u=0 m/s at the beginning of the simulation, at the initial values from which the transient solver starts to compute the solution.

This is referred to as Inconsistent Initial Values.

COMSOL will, by default, attempt to reconcile inconsistent initial values by taking a small time step that is a fraction of the initial time step using a Backward Euler method. By default, the initial time step is automatically determined based upon the total simulation time span, but it is possible to manually set the initial time step and also to change the fraction of that initial step used by the Backward Euler method for the consistent initialization. These settings will affect the results of the consistent initialization procedure. The consistent initialization may fail if the Backward Euler step is too large or the boundary conditions truly are inconsistent. More commonly the results may be very far from what is expected, and the solver will likely need to take very small time steps at the beginning of the simulation to evolve from this solution. This is undesirable as it leads to relatively long solution times.

Self check 4	Written test.

Direction: write the answer for the following questions. Use the Answer sheet provided

1. What is the common problem or mistake in flow condition?
2. COMSOL will, by default, attempt to reconcile inconsistent initial values by taking a small time step that is a fraction of the initial time step using a ______________________.
3. What settings will affect the results of the consistent initialization procedure and in what case it fail?

Note: **Satisfactory rating - 5 points** **Unsatisfactory - below 5 points**

You can ask you teacher for the copy of the correct answers.

Answer sheet

Name: _________________Date: ______________

Answer

1. ____
2. ____
3. ____

Instruction Sheet-2	Learning guide 43: Calculate hydraulic and energy gradient for pipeline

This learning guide is developed to provide you the necessary information regarding the following **content coverage** and topics:

- Standard formulae
- Pipeline design charts
- Roughness coefficients
- Calculating pipe line system using hydraulic gradient line
- Calculating pipe discharges

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, **upon completion of this Learning Guide, you will be able to**:

- .Measurements are reviewed and compared against expected trends.
- Standard processes and software are used to check, edit, verify and audit data.
- Standard processes are used to identify, estimate, adjust and justify data and review inconsistent data on flow conditions.
- Records are prepared in a format suitable for dissemination

Learning Instructions:

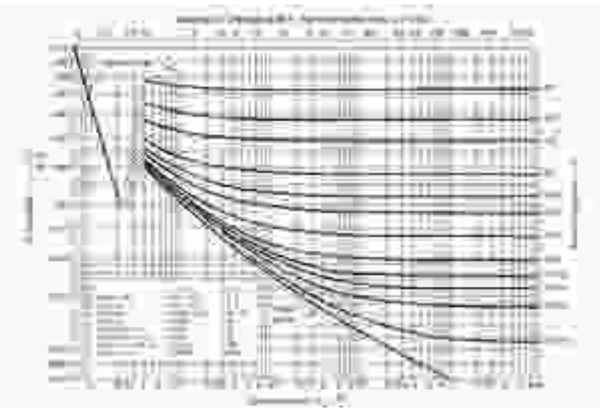
1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below
3. Read the information written in the “Information Sheets 1- 5”. Try to understand what are being discussed.
4. Accomplish the “Self-checks1,2,3,4and 5 ” in each information sheets on pages 39, 44, 49, 54 and 60
5. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-checks).
6. After You accomplish self check, ensure you have a formative assessment and get a satisfactory result; then proceed to the next LG.

Information Sheet-	Standard formula

Calculating Major Losses Using Different Formulas

- **Darcy Weisbach Formula -**

$$hf = \frac{4fLV^2}{2gd}$$



Moody's Diagram

Where,

hf = loss of head due to friction,

f = coefficient of friction which is function of Re number, derived from Moody's Diagram,

L = length of pipe,

V = mean velocity of pipe,

d = diameter of pipe.

- **Chezy's Formula -**

V=C √mi

Where,

C = Chezy's constant,

m = hydraulic mean depth (Area / Perimeter),

i = hf / L, which is head loss of head per unit length of pipe.

- **Hazen-Williams Formula -**

$$h = \frac{kL}{D^{1.16}}\left(\frac{V}{C}\right)^{1.85}$$

Where,

h = head loss (m)

D = diameter (m)

V = velocity (m/s)

C = Hazen-Williams C-factor

L = length (m)

k = 6.79 for V in m/s, D in m or

The C-factor ranges between 0 to 150 depending on the material and age of the pipe.

- **Modified Hazen-Williams Formula -**

$$V = 143.534\ Cr\ r^{0.6575} S^{0.5525}$$

$$h = [L(Q / Cr)^{1.81}]/994.62D^{4.81}$$

where,

V = velocity of flow in m/s,

Cr = pipe roughness coefficient, (1 for smooth pipes; < 1 for rough pipes),

r = hydraulic radius in m,

S = friction slope,

h = friction head loss in m.

Difference between the Darcy Weisbach and Hazen-Williams formulas:

As a part of academic curriculum, Darcy Weisbach formula has enjoyed much popularity as compared to any other formula owning to extreme precision in the values of the resulting head loss.

But, in practice, the engineers are challenged with high time consumption in determining the value of f - coefficient of friction for every pipe using Moody's Diagram. Because of this reason, its popularity among practitioners is very less.

Hazen-Williams formula, perhaps, generates reliable and reasonably precise values of head loss. The Hazen Williams C-factor ranges between 0 to 150. Rougher the pipe is, smaller would be the value of C and smoother the pipe is, higher would be the value of C.

Self-Check - 1	**Written Test**

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. Try to draw moody diagram.
2. Write the Modified Hazen-Williams Formula.
3. What is Difference between the Darcy Weisbach and Hazen-Williams formulas.

Answer sheet

Note: **Satisfactory rating - 5 points** **Unsatisfactory - below 5points**

You can ask you teacher for the copy of the correct answers.

Answers

7. ____

8. ____

9. ____

Information Sheet- 2	Pipeline design charts

INTRODUCTION TO PIPING SYSTEM

What is piping systems? the piping system install to convey the fluids required for chemical processes or otherwise between the various equipment and end users and consist of various components such as valves, fittings, online measuring instruments, etc. is called as a 'Piping Systems'

PIPING COMPONENTS

Mechanical element suitable for joining or assembly in to pressure-tight fluid containing piping systems. Components include pipe, fittings, flanges, gaskets, bolting, valves and devices such as expansion joint, flexible joints, pressure hoses, traps, strainers, in line portions of instruments, and separator

Pipe

A pipe or tube is hollow, longitudinal product. ' A tube' is a general term used for hollow product having circular, elliptical or square cross section or for that matter cross section of any perimeter. A pipe is tubular product of circular cross-section that has specific sizes and thickness governed by particular dimensional standard

Classifications

Pipe can be classified based on method of manufacture or based on their applications

Method of Manufacture

Seamless pipes are manufactured by drawing or extrusion process

ERW pipes (Electric resistance welded pipes) are formed from a strip which is longitudinally welded along its length. Welding may be by electric resistance, high frequency, or induction welding. ERW pipes can also be drawn for obtaining required dimensions and tolerances.

Pipes in small quantities are manufactured by EFW (Electric fusion welding) Process where in instead of electric resistance welding, the longitudinal seam is welded by manual or automatic electric arc process.

There are spiral seam welded pipes, which are large dia pipes 500 NB and above. And pipes are made by welding a spiral seam produced by forming continues steel skelp in to circular shape

Centrifugally cast pipes are made by spraying molten metal along a rotating die where the pipes are cast in shape due to centrifugal action

Classification based on Applications

Pipes are classified as:

- Pressure Pipes or Process pipes
- Line Pipes
- Structural Pipes

Pressure pipes are those which are subjected to fluid pressure and or temperatures. Fluid pressure in generally internal pressure due to fluid being conveyed or may be external pressure (i.e. jacketed piping) and are mainly used as plant piping.

Line pipes are mainly used for conveying of the fluid and are subjected to fluid pressure. These are generally not subjected to high temperatures.

Structural pipes are not used for conveying fluids and therefore not subjected to fluid pressures or temperatures. They are used as structural components (e.g. handrails, columns, sleeves etc) and are subjected to static load only.

Pipes Dimensional Standards (ASME B 36.10, ASME B 36.19)

Diameters: Pipes are designated by Nominal size, starting from 1/8” Nominal size, and increasing in step

1 Pipes sizes increases in steps of 1/8” to ½” =1/8”,1/4”,3/8”,1/2”, Nominal size

2 Sizes in step of ¼” = ½”,3/4”,1”,1 ¼”,1 ½”

3 In step of ½” up to 4” = 1 ½”, 2”, 2 ½”, 3”, 3 ½”,4”.

4 In step of 1” up to 6” = 4”,5”,6”.

5 In step of 2” up to 36” = 6”,8”,10” ………Etc

Types of pipe

Table 1. Heat Pipe Design Requirements.

Working Fluid	Water
Nominal Operating Temperature	500K (227°C)
Operating Temperature Range	310K to 550K (37 to 277°C)
Heat Pipe Heat Transfer Capability	Maximize at 500K
Heat Pipe Outer Diameter	1.27 cm (0.50")
Heat Pipe Evaporator Length	25 cm (9.84")
Heat Pipe Condenser Length	90 cm (35.43")
Heat Pipe Mass	Minimize
Wick Structure Type & Material	Solid Titanium Axial Grooves
Heat Pipe Closure	Titanium Swagelok Bellows Valve

Fixture	Minimum Pipe Size (in.)	Flow Rate (GPM)	Pressure (psi)
Bathtub	1/2"	4	8
Dishwasher	1/2"	2.75	8
Drinking Fountain	5/8"	0.75	8
Hose Bibb	1/2" Int, 3/4" Ext	5, 15	8, 15
Kitchen Sink	1/2"	2.5	8
Laundry	1/2"	4	8
Lavatory	3/8"	2	8
Shower	1/2"	3	8
Service Sink	1/2"	3	8
Urinal Flush Valve	3/4"	1.6	15
Water Closet Tank	3/8"	1.6	15
Water Closet Valve	1"	1.6	15

Table 2 types of pipe fixtures

Pipeline design sample problems

Plotting the pipeline profile is a process of trial and error. The calculated values for head losses from different sizes of pipe are compared to the available head on the profile drawing. The smallest diameter pipe that results in acceptable flow and pressure is chosen for each continuous section of the pipeline.

Example 1: A spring with a flow of 0.5 L/s is 1,000 meters from the farmer's field, and the available head is 20 meters. It is planned to convey the entire flow to a small reservoir. What size pipe is recommended?

Add 10% additional friction losses to the friction losses estimated from using Tables such as 5.3 and 5.4 or the Hazen-Williams equation (see section 5.2.8, "Calculating Friction Losses"). Using Table 5.4 with an additional 10% friction loss and assuming the use of galvanized iron pipe, a flow of 0.5 L/s, and a length of 1,000 meters, a 1.5-inch pipe results in a head loss of 11 meters. For 1.25-inch pipe, the head loss is 26 meters. Thus, the required flow will not be obtained with a 1.25-inch pipe. A 1.5-inch pipe could be used; however, the most economical solution is a combination of two pipe sizes. A 1.5-inch GI pipeline of 500 meter length with a flow of 0.5 L/s has a head loss of 6 meters, and 500 meters of 1.25-inch GI pipe has a head loss of 13 meters. Thus, the total head loss for the 1,000 meter pipeline is 19 meters, which closely matches the available head. The pipeline profile and HGL are plotted in Figure 2.1.

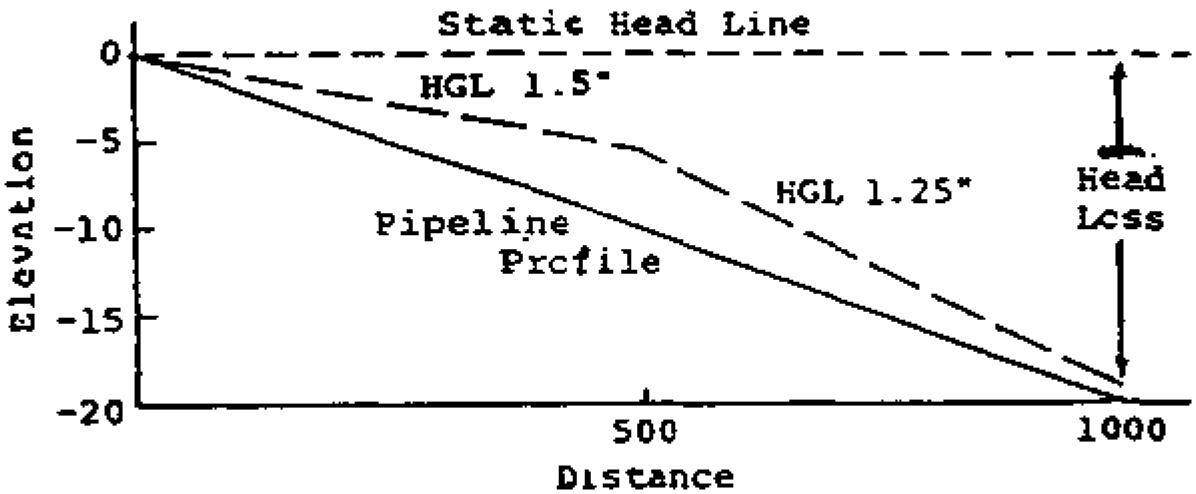


Figure 2 1 Example: Sketch for Pipeline 1 (Ref. 21)

Self-Check -2	Written Test

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1;-Explain about pipe flow?

2;-What is the standard pipe flow?

3;-write Types of pipes?

4;-write the fixture of pipe design factor?

Answer sheet

Note: **Satisfactory rating - 5 points** **Unsatisfactory - below 5points**

You can ask you teacher for the copy of the correct answers.

Answers

1. ____
____
____

2. ____
____
____

3. ____
____
____

4. ____
____
____

Information Sheet- 3	Roughness coefficient

Introduction

The resistance to flow in open channels depends on many flow and channel parameters. Out of the many factors, vegetation is the most important parameter in vegetative channels. Vegetation in an open channel retards the water flow by causing energy loss through turbulence and by exerting additional drag forces on the moving liquid. Presence of vegetation in a channel modify the velocity profiles, and hence the resistance in terms of roughness coefficients. The roughness coefficients of such channels change with the flow depths and from sections to sections. Because of this complex nature, it is hard to develop a flow model based on theoretical calculations and derivations. A laboratory study to explore the effect of vegetation in terms of rigid cylindrical roughness on the behaviors of Manning's roughness coefficient n, Chezy's coefficient C and Darcy-Weisbach's friction factor f in an open channel is presented. The study consists of flume experiments for flows with unsubmerged rigid cylindrical stems of a concentration and diameter arranged in a regular staggered configuration. The usual practice in 1D analysis is to select a value of n depending on the channel surface roughness and take it as uniform for the entire surface for all depths of flow. The influences of all the parameters are assumed to be lumped into a single value of n, C and f. Researches have shown that the coefficients not only denote the roughness characteristics of a channel but also the energy loss in the flow. The larger the value of n, the higher is the loss of energy within the flow Different roughness coefficients are found to vary differently with the non-dimensional hydraulic, geometric and surface parameters. Behaviours of different resistance coefficients due to vegetation are discussed and results are summarized and presented. Graphs of aspect ratio vs. n, C and f respectively are presented

Manning's roughness coefficient

The Manning's roughness coefficient is used in the Manning's formula to calculate flow in open channels.

Coefficients for some commonly used surface materials:

Surface Material	Manning's Roughness Coefficient - *n* -
Asbestos cement	0.011
Asphalt	0.016
Brass	0.011
Brick and cement mortar sewers	0.015
Canvas	0.012
Cast or Ductile iron, new	0.012
Clay tile	0.014
Concrete - steel forms	0.011
Concrete (Cement) - finished	0.012
Concrete - wooden forms	0.015
Concrete - centrifugally spun	0.013
Copper	0.011
Corrugated metal	0.022
Earth, smooth	0.018
Earth channel - clean	0.022
Earth channel - gravelly	0.025
Earth channel - weedy	0.030
Earth channel - stony, cobbles	0.035
Floodplains - pasture, farmland	0.035
Floodplains - light brush	0.050
Floodplains - heavy brush	0.075
Floodplains - trees	0.15

Surface Material	Manning's Roughness Coefficient *- n -*
Galvanized iron	0.016
Glass	0.010
Gravel, firm	0.023
Lead	0.011
Masonry	0.025
Metal - corrugated	0.022
Natural streams - clean and straight	0.030
Natural streams - major rivers	0.035
Natural streams - sluggish with deep pools	0.040
Natural channels, very poor condition	0.060
Plastic	0.009
Polyethylene PE - Corrugated with smooth inner walls	0.009 - 0.015
Polyethylene PE - Corrugated with corrugated inner walls	0.018 - 0.025
Polyvinyl Chloride PVC - with smooth inner walls	0.009 - 0.011
Rubble Masonry	0.017 - 0.022
Steel - Coal-tar enamel	0.010
Steel - smooth	0.012
Steel - New unlined	0.011
Steel - Riveted	0.019
Vitrified clay sewer pipe	0.013 - 0.015
Wood - planed	0.012
Wood - unplaned	0.013

Surface Material	Manning's Roughness Coefficient - *n* -
Wood stave pipe, small diameter	0.011 - 0.012
Wood stave pipe, large diameter	0.012 - 0.013

Self-Check -3	Written Test

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. What is roughness coefficient?
2. What are factors that affect roughness coefficient?

Note: **Satisfactory rating – 3 and above points** **Unsatisfactory - below 3 points**

You can ask you teacher for the copy of the correct answers.

Answer Sheet-5

Name: _________________________ Date: _______________

Short answer

1. ____
____
____
____
2. ____
____
____
_____________________________.

Information Sheet-4	Calculating hydraulic gradient line

Introduction

In the flow of a fluid through pipes, it is seen that there is a loss of head. Whilst some of this is due to the effect of sudden contraction or expansions in the pipe diameter, pipe fittings such as bends and valves and entry and exit losses, a loss of potential head (i.e. The input of the pipe is higher than the outflow) a significant portion is due to the friction in the pipe (The Darcy Equation). However in a pipe of uniform cross section, there will be no loss of velocity head and so the loss of **Total Energy** will be the result of a loss in **Pressure Head**.

The following diagram shows a uniform pipe and the value of pressure head at three points down its length. The line joining these points is called **The Hydraulic Gradient**

A **pipe** is a tubular section or hollow cylinder, usually but not necessarily of circular cross-section, used mainly to convey substances which can flow-liquids and gases (fluids), slurries, powders, masses of small solids.

The **hydraulic gradient** is a vector gradient between two or more hydraulic head measurements over the length of the flow path.

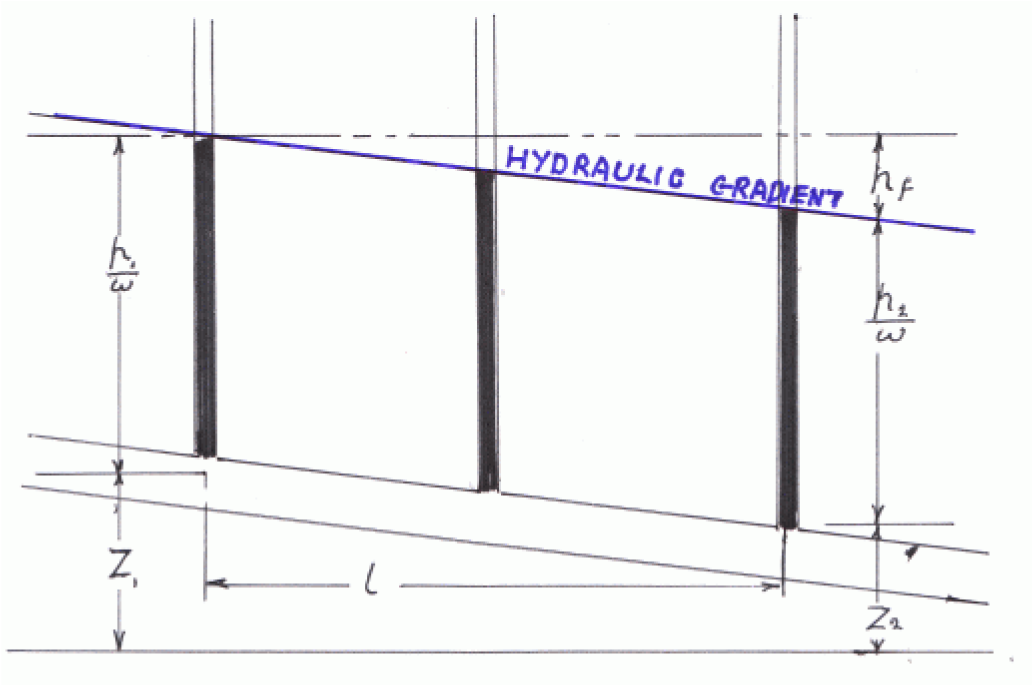


figure 4.1 Hydraulic Gradient

The Significance Of The Hydraulic Gradient

Normally when a pipe is laid, attempts are made to keep the pipe at or below the hydraulic gradient. However in some cases this may not be possible but provided that the pipe does not rise by more than about 26 ft. (8 mtr.) water will still flow. Above this height air comes out of solution and an airlock is formed.

The Hydraulic Gradient

The above diagram is of an extremely simple system. On the next diagram the pipe has both a sudden contraction and an enlargement. The various losses in energy are shown and this is used to construct the **Total Energy Line** which is shown in red on the diagram.

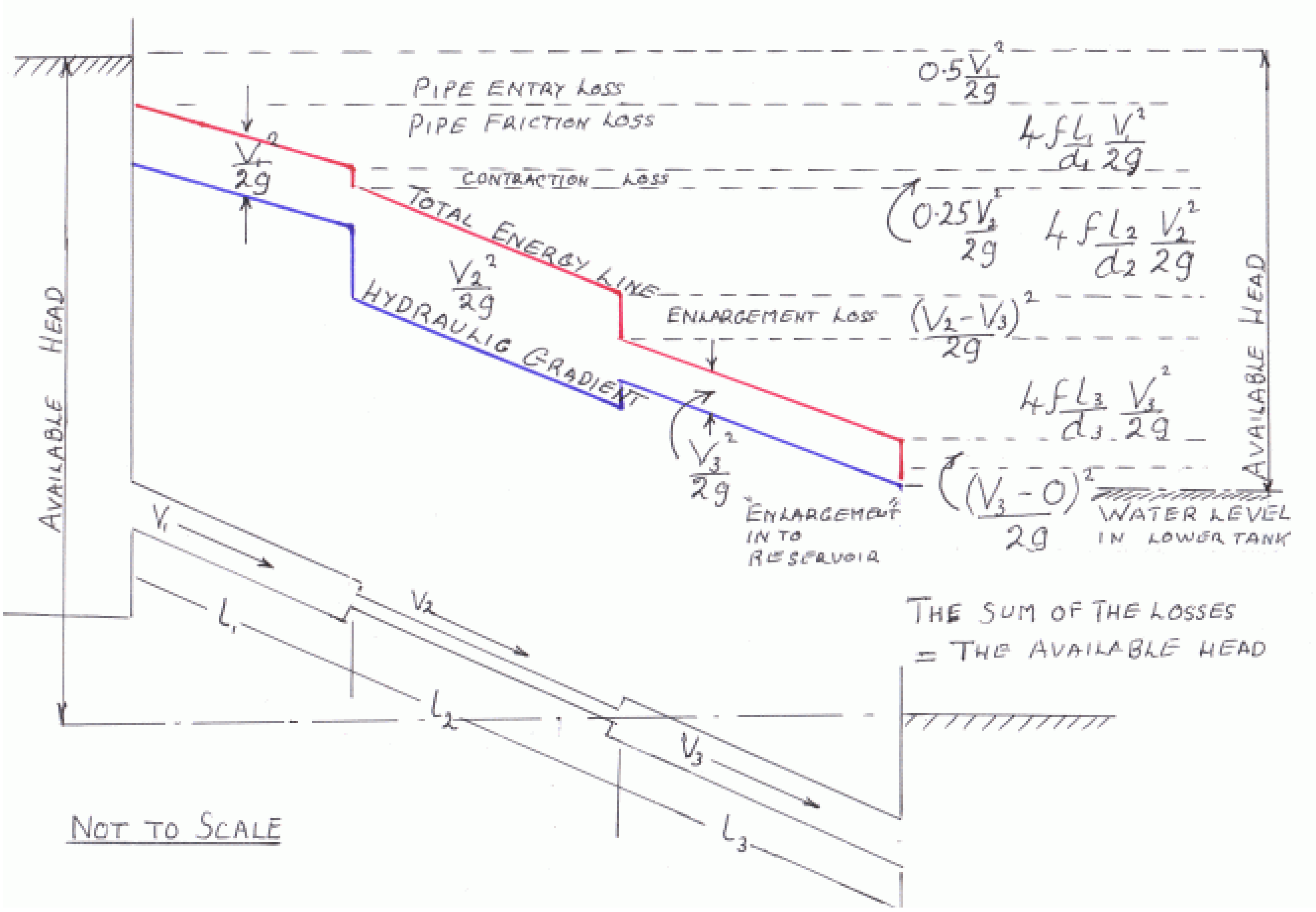


figure 4.2 water level

The velocity head at the salient points in the pipe are also calculated and these are subtracted from the energy line to give the **Hydraulic Gradient** which is shown in blue.

Example:

[imperial]

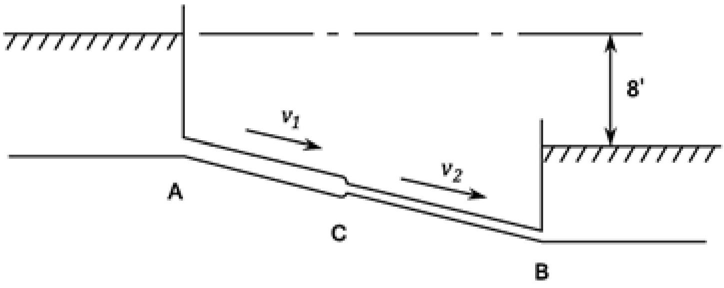
Example - Example 1

Problem

Two tanks and are connected by a pipe 100 ft. long.

The first 70 ft. has a diameter of 3 in. and then the pipe is suddenly reduced to 2 in. for the remaining 30 ft. The difference of levels between the tanks is constant at 30 ft. = 0.005 and the coefficient of contraction at all sudden changes of area is 0.58.

Find all the **head losses** including that at the sharp edged pipe entry at in terms of the velocity and hence find the flow in gallons/min. Draw in **Hydraulic Gradient diagram**.



Workings From the diagram: Substituting given values: And:

- The loss at the pipe entry
- The frictional loss along (The **Darcy equation**):
- The loss caused by the sudden contraction at :
- The frictional loss along :
- The velocity loss at the exit into :

It can be seen from the diagram that the total head lost is 8 ft. and therefore:

Hence:

And:

The flow through the pipes is the product of the cross sectional area of the pipe and the velocity. i.e.

The flow is:

Note that 1 of water weighs 62.4 lb. and 1 English gallon weighs 10 lbs.

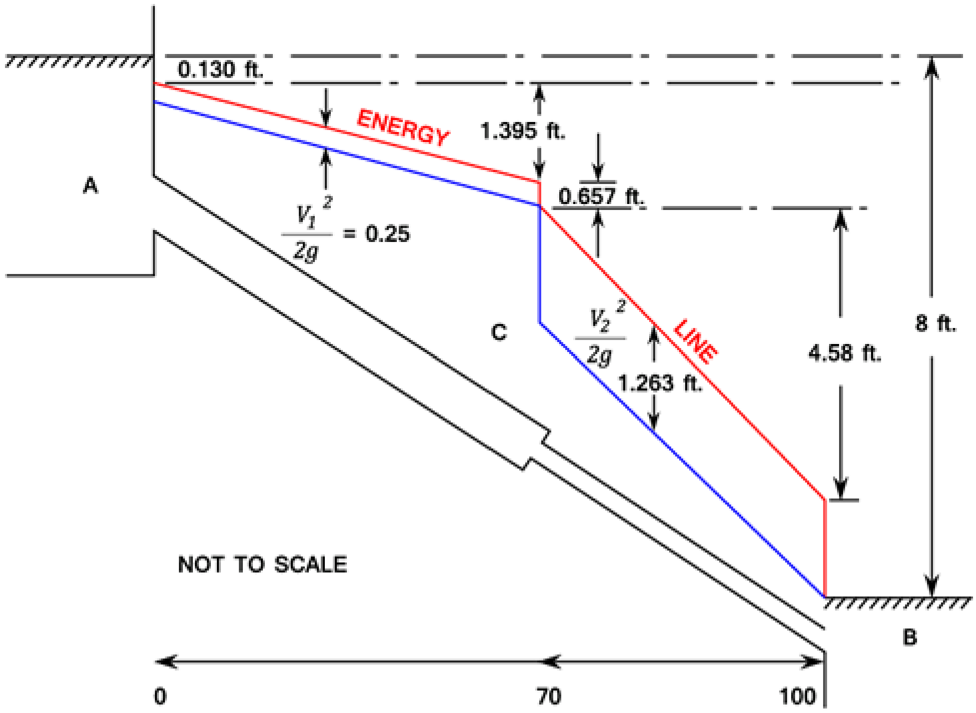
The Hydraulic Gradient

1. The velocity head in :

- The velocity head in :

The various head losses have already been written down in terms of

It is thus now possible to tabulate actual values and enter them on a diagram of the two reservoirs and the connecting pipes.



The Hydraulic Gradient is the line shown in Blue

Solution

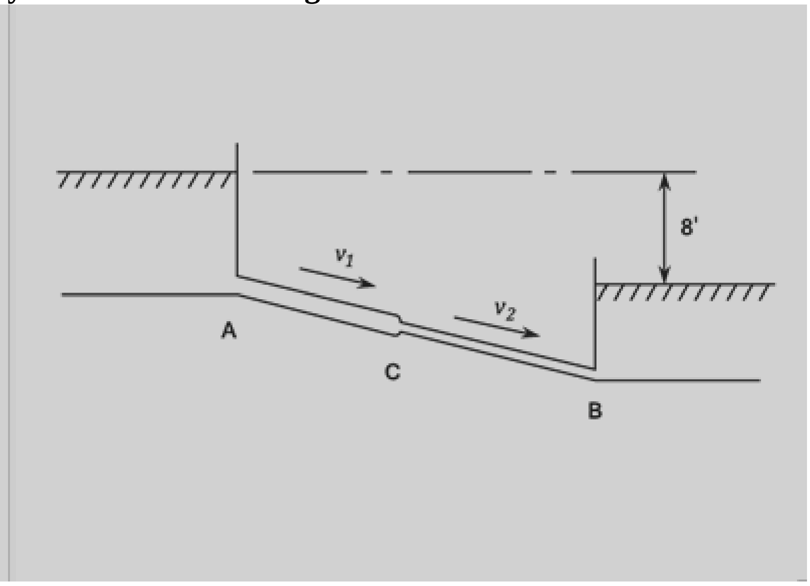
The losses are:

- **Entry at :** 0.130 ft.
- **Pipe friction:** 1.395 ft.
- **Loss at :** 0.657 ft.
- **Pipe Friction:** 4.550 ft.
- **Exit loss:** 1.263 ft.

Self-Check -4	Written Test

Direction : calculate the given question and give the answer.

1. Two tanks and are connected by a pipe 100 ft. long. The first 70 ft. has a diameter of 3 in. and then the pipe is suddenly reduced to 2 in. for the remaining 30 ft. The difference of levels between the tanks is constant at 30 ft. = 0.005 and the coefficient of contraction at all sudden changes of area is 0.58. Find all the **head losses** including that at the sharp edged pipe entry at in terms of the velocity and hence find the flow in gallons/min. Draw in **Hydraulic Gradient diagram**.



channel flow: Uniform flow, best hydraulic sections, energy principles, Froude number

Open channel flow must have a free surface. Normally free water surface is subjected to atmospheric pressure, which remains relatively constant throughout the entire length of the channel. In free-surface flow, the component of the weight of water in the downstream direction causes acceleration of flow (it causes deceleration if the bottom slope is negative), whereas the shear stress at the channel bottom and sides offers resistance to flow. Depending upon the relative magnitude of these accelerating and decelerating forces, the flow may accelerate or decelerate. For example, if the resistive force is more than the component of the weight, then the flow velocity decreases and, to satisfy the continuity equation, the flow depth increases.The converse is true if the component of the weight is more than the resistive force. However, if the channel is long and prismatic(i.e., channel cross section and bottom slope do not change with distance), then the flow accelerates or decelerates for a distance until the accelerating and resistive forces are equal. From that point on, the flow velocity and flow depth remain constant Such a flow, in which the flow depth does not change with distance, is called uniform flow, and the corresponding flow depth is called the normal depth.Uniform flow is discussed in this chapter. An equation relating the bottom shear stress to different flow variables is first derived. Various empirical resistance formulas used for the free-surface flows are then presented. A procedure for computing the normal depth for a specified discharge in a channel of known properties is outlined.

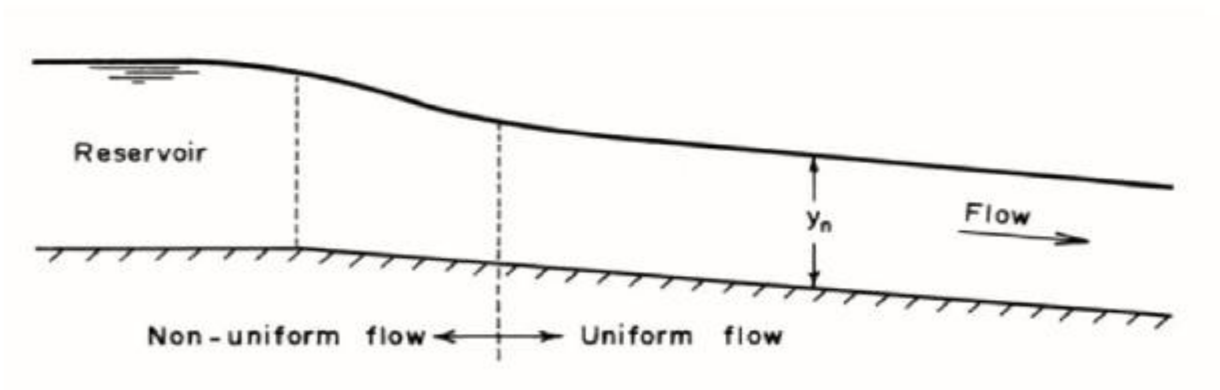


Figure 5.1 flowing diagram

Manning Equation

Since the derivation of the Chezy equation in 1768, several researchers have tried to develop a rational procedure for estimating the value of Chezy constant, C. However, unlike the Darcy-Weisbach friction factor for the closed conduits, these attempts have not been very successful, because C depends upon several parameters in addition to the channel roughness.

$$C \propto R^{1/6}$$

French engineer named A. Flamant incorrectly attributed the above equation to an Irishman, R. Manning, and expressed it in the followingform in 1891

$$V = 1/nR^{2/3}S^{1/2}$$

CHEZY'S EQUATION

$$V = C\sqrt{RSo}$$

Pitot Tube

A pitot tube is a pressure measurement instrument used to measurefluidflow velocity. The pitot tube was invented by theFrenchengineerHenri Pitotin the early 18th centuryand was modified to its modern form in the mid-19th century by French scientistHenry Darcy.¬It is widely used to determine theairspeedof anaircraft, water speed of a boat, and to measure liquid, air and gas flow velocities in industrial applications.¬The pitot tube is used to measure the local flow velocity at a given point in the flow stream and not the average flow velocity in the pipe or conduit.¬The basic pitot tube consists of a tube pointing directly into the fluid flow. As this tube contains fluid, a pressure can be measured; the moving fluid is brought to rest (stagnates) as there is no outlet to allow flow to continue. ¬This pressure is the stagnation pressureof the fluid, also known as the total pressure or (particularly in aviation) the pitot pressure.¬The measured stagnation pressure cannot itself beused to determine the fluid flow velocity (airspeed in aviation). However,Bernoulli'sstates:¬Stagnation pressure = static pressure + dynamic pressure

CURRENT METER

Acurrent meterisoceanographicdevice forflow measurementby mechanical (rotor current meter), tilt (Tilt Current Meter), acoustical (ADCP) or electrical means.

MEASUREMENT PRINCIPLES

a. Mechanical current meters are mostly based on counting the rotations of a propeller and are thusrotor current meters. A mid-20th-century realization is theEkman current meterwhich drops balls into a container to count the number of rotations.

b.**Acoustic** Thereare two basic types of acoustic current meters: Doppler and Travel Time. Both methods use a ceramic transducer to emit a sound into the water.Doppler instruments are more common.An instrument of this type is theAcoustic Doppler Current Profiler(ADCP)which measures thewater currentvelocitiesover a depth range using theDoppler effectofsound wavesscattered back from particles within the water column. The ADCPs use the traveling time of the sound to determine the position of the moving particles. Single-point devices use again the Doppler shift, but ignoring the traveling times. Such a single point Doppler Current Sensor (DCS) has a typical velocity range of 0 to 300cm/s.Travel time instruments determine water velocity by at least two acoustic signals, one up stream and one down stream.

c. Electromagnetic Induction This novel approach is for instance employed in the Florida Strait where electromagnetic induction in submerged telephone cableis used to estimate the through-flow through the gatewayand the complete setup can be seen asone huge current meter. it is possible to evaluate the variability of the averaged horizontal flow by measuring the induced electric currents. The method has a minor vertical weighting effect due to small conductivity changes at different depths.

d.Tilt. current meters operate under the drag-tilt principle. They consist of a sub-surface buoy that is anchored to the sea floor with a flexible line or tether. The float tilts as a function of its shape, buoyancy and the water velocity. Once the characteristics of a given buoy are known, the velocity can be determined by measuring the angle of the buoy.A Tilt Current Meter is typically deployed on the bottom with an anchor but may be deployed on lobster traps or other convenient anchors of opportunity.

VENTURIMETER

Venturimeter is a device used for measuring the rate of flow of a fluid flowing through a pipe. It consists of three parts:• A short converging part

Throat Diverging partLet d1 = diameter at the inlet (section 1)p1 = pressure at section 1v1 = velocity at section 1A1= area at section1 d2, p2, v2, A2 are the corresponding values at the throat (section 2) Applying Bernoulli's equations at sections 1 and 2, we get

ORIFICE METER

An orifice meter is a conduit and a restriction to create a pressure drop. An hour glass is a form of orifice. A nozzle, venturi or thin sharp edged orifice can be used as the flow restriction. In order to use any of these devices for measurement it is necessary to empirically calibrate them.

That is, pass a known volume through the meter and note the reading in order to provide a standard for measuring other quantities. Due to the ease of duplicating and the simple construction, the thin sharp edged orifice has been adopted as a standard and extensive calibration work has been done so that it is widely accepted as a standard means of measuring fluids.

WEIRS:

A weir is a barrier across a river designed to alter its flow characteristics. In most cases, weirs take the form of obstructions smaller than most conventional dams, pooling water behind them while also allowing it to flow steadily over their tops. Weirs are commonly used to alter the flow of rivers to prevent flooding, measure discharge, and help render rivers navigable.

FUNCTIONS:

Weirs allow hydrologists and engineers a simple method of measuring the volumetric flow rate in small to medium-sized streams or in industrial discharge locations. Since the geometry of the top of the weir is known and all water flows over the weir, the depth of water behind the weir can be converted to a rate of flow. The calculation relies on the fact that fluid will pass through the critical depth of the flow regime in the vicinity of the crest of the weir.

TYPES OF WEIRS

Broad-crested weirA broad-crested weir is a flat-crested structure, with a long crest compared to the flow thickness. When the crest is "broad", the streamlines become parallel to the crest invert and the pressure distribution above the crest is hydrostatic.

Sharp crested weir

A sharp-crested weir allows the water to fall cleanly away from the weir. Sharp crested weirs are typically1⁄4inch (6.4mm) or thinner metal plates. Sharp crested weirs come in many different shapes and styles, such as rectangular (with and without end contractions),Compound weirThe sharp crested weirs can be consolidated into three geometrical groups: a) The rectangular weirb) The V or triangular notchc) Special notches, such as trapezoidal, circular or parabolic weirs. For accurate flow measurement over a wider range of flow rates, a compound weir combines two or more types -typically a V-notch weir with a rectangular weir.

RECTANGULAR NOTCH

Q=Ce√b)(h+Kh)3/2WhereQ= DischargeCe =Discharge Coefficientg= Acceleration due to gravityb =Notch widthh=HeadKband Kh accounts for effects of viscosity and surface tension

Self check 5	Written test.

Direction: write the answer for the following questions. Use the Answer sheet provided

1. What is open channel flow?
2. What is manning equation?
3. What are the measurement principles of flow discharge

Note: **Satisfactory rating - 5 points** **Unsatisfactory - below 5 points**

You can ask you teacher for the copy of the correct answers.

Answer sheet

Name: _________________Date: ______________

Answer

1. ____
2. ____
3. ____

Instruction Sheet 3	Learning Guide 44: Calculate flow in open channels

This learning guide is developed to provide you the necessary information regarding the following **content coverage** and topics:

- Hydraulic principles
- Hydraulic flows
- Types of flow condition
- Methods used for measuring flows
- formulae for calculating flows
- Open channel

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, **upon completion of this Learning Guide, you will be able to**:

- .Identifying *methods used for measuring flows* in open channels
- Using the *formulae for calculating flows* in open channels
- Distinguishing the *characteristics of open channels*
- Use of different measuring instruments and devices used in open channels
- The *hydraulic principles* which apply to different *meters* are assessed.
- The limitations of the meters are identified.

Learning Instructions:

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below
3. Read the information written in the “Information Sheets 1- 6”. Try to understand what are being discussed.
4. Accomplish the “Self-checks1,2,3,4 ,5 and 6 ” in each information sheets on pages 4,11,14,19,23 and 26.
5. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-checks).
6. After You accomplish selchecksf, ensure you have a formative assessment and get a satisfactory result; then proceed to the next LG.

Information Sheet 1	Hydraulics

Introduction

Hydraulic system, from a general perspective, is an arrangement of interconnected components that uses a liquid under pressure to provide energy transmission and control. It has an extremely broad range of applications covering basically all elds of production, manufacturing and service. Consequently, the energy transmission and control requirements are very diverse and thus the structure of each hydraulic system has its specicities.However, on analyzing the current hydraulic systems, one can identify four main functions [1], as presented in Figure 1.1, which are: primary energy conversion, energy limitation and control, secondary energy conversion, and -uid storage and conditioning.Furthermore, this gure shows the main resources that -ow through a hydraulic system and which can be grouped into the classes: information, material, and energy [2].The input of mechanical energy (M), which is a result of the external conversion of primary electrical or chemical (combustion) energy, is converted into hydraulic energy (H). Using signals or data (S, D) from an operator or from other equipment, the hydraulic energy (H) is limited and con-trolled such that it becomes appropriate for conversion into mechanical energy (M). This mechanical energy is the desired output of the hydraulic system and will be used to drive or move external devices.The hydraulic energy is carried by the hydraulic -uid (F) and thus its storage and conditioning, including contamination and temperature control, are also essential functions.As a consequence of the physical phenomena, construction characteristics, and circuit arrange-ment, part of the useful energy is dissipated in a hydraulic system. Therefore, all functions transfer thermal energy (T) to the -uid and to the environment.

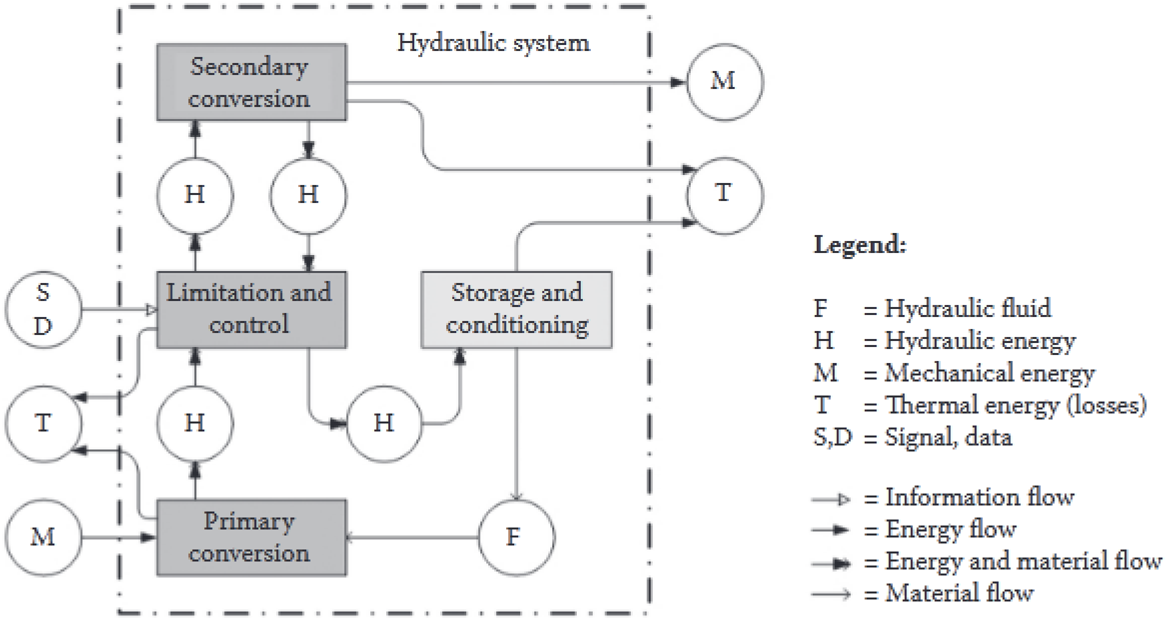


FIGURE 1.1 Generic hydraulic system: Functions and resource flows.

Self-Check -1	Written Test

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. Explain about open channel?
2. What is the difference b/n pipe and channel?
3. What is hydraulic jump ?

Part two Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page

1. Write the difrence b/n water haydrolic?

2. Define haydrolic?

Information Sheet 2	Hydraulics Flow

Flow classification

Recalling that flow may be steady or unsteady and uniform or non-uniform, the major classifications applied to open channels are as follows:

Steady and Unsteady: Time is the criterion. Flow is said to be steady if the depth of flow at a particular point does not change or can be considered constant for the time interval under consideration. The flow is unsteady if depth changes with time.

Uniform and non-uniform Flow: Space as the criterion. Open Channel flow is said to be uniform if the depth and velocity of flow are the same at every section of the channel. Hence it follows that uniform flow can only occur in prismatic channels. Flow in channels is termed as non-uniform or varied if the depth of flow y, changes from section to section

$$\left(\frac{\partial y}{\partial s}\right) \neq 0 .$$

Non-uniform flow is rapidly varied flow If the depth of flow changes a abruptly over a comparatively short distance; eg. Hydraulic Jump Non-Uniform flow is gradually varied flow. If the change in depth of flow takes place gradually in a long reach of the channel.

Steady uniform flow, in which the depth is constant, both with time and distance. This constitutes the fundamental type of flow in an open channel in which the gravity forces are in equilibrium with the resistance forces.

Steady non-uniform flow, in which the depth varies with distance, but not with time. The flow may be either (a) gradually varied or (b) rapidly varied.Type (a) requires the joint application of energy and frictional resistance equations. Type (b) requires the application of energy and momentum principles.

Unsteady non uniform flow, in which the depth varies with both time and distance (unsteady uniform flow is very rare). This is the most complex flow type, requiring the solution of energy, momentum and friction equations through time. The various flow types are all shown in Figure 1.2.

Laminar and turbulent flow

Flow is said to be laminar when adjacent fluid layer, move at the same (or nearlythe same) velocity and paths of individual fluid particles do not cross or intersect.

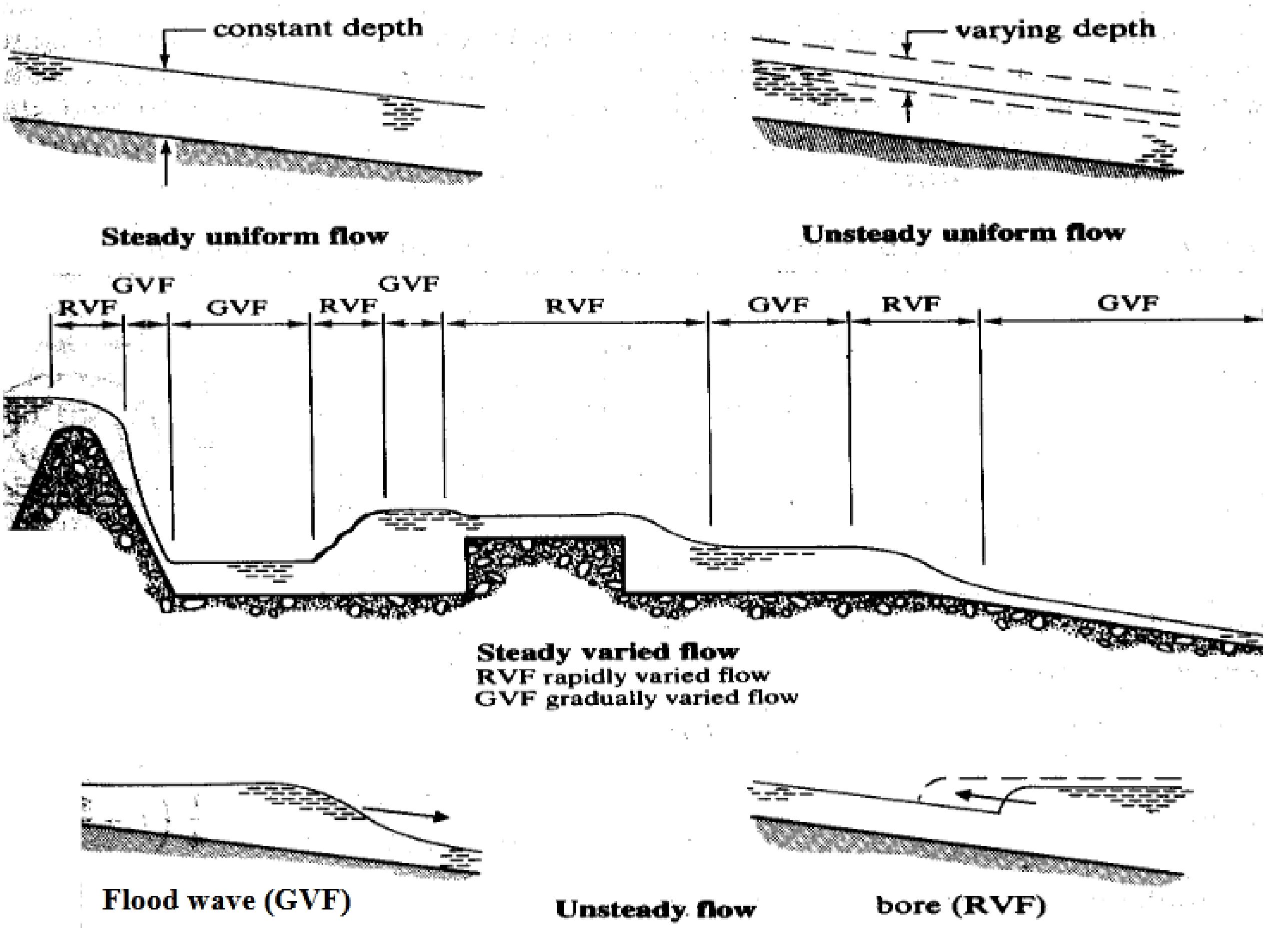


Fig. 1.2 types of flow

$$R_e = \frac{\rho V R}{\mu}, \textit{for open channel}$$

$$R_e = \frac{\rho D V}{\mu}, \textit{ for closed channel}$$

Where R_e = Reynolds number

ρ = mass density of the fluid

D = Pipe diameter

V = Velocity of the fluid

μ = dynamic viscosity of the fluid

R = hydraulic mean radius

R = A/p , where A – area of cross section of the channel

P – Wetted perimeter of the channel.

On this experimental data it has been found that

If RV 500 $\leq$ Re $\leq$ 600, the flow is considered laminar

If Re > 2000, the flow is considered turbulent

If 500 $\leq$ Re $\leq$ 2000, the flow is transition state.

Critical and supercritical flow Gravity is a predominant force in the case of channel flow. As such depending on the relative effect of gravity the inertia forces, the channel flow may be designed as sub critical, critical or super critical. Determination of such flow depend on the dimensionless parameter called fraud number (Fr) which is defined as the ratio of inertia the gravity forces.

$Fr = \frac{V}{\sqrt{gy}}$, Where V = the mean velocity of flow

g = acceleration due to gravity

y = hydraulic depth of the channel

y = A/T, T = top width, A = cross sectional area

This experiment indicate that

When Fr = 1, the channel flow is said to be critical state

If Fr < 1, or $V < \sqrt{gD}$, the flow is sub critical (tranquil or streaming)

If Fr > 1, or $V > \sqrt{gD}$, the flow is said to be supercritical or rapid or shooting or torrential flow.

Self-Check -2	Written Test

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

7. What are the classification of open channel?
8. Write the equation for fraud number?
9. What are equation used to calculate Reynolds number ?
   A. In open channel
   B. In cloth channel

Information Sheet 3	Methods used for measuring flows

Flow Measurement In Open Channels

Open channel flow is flow in any channel in which the liquid flows with a free surface. Included are tunnels, partially filled pipes, canals, streams, and rivers. There are many methods of determining the rate of flow in open channels. Some of the more common include the timed gravimetric, dilution, velocity-area, hydraulic structures, and slope-hydraulic radius-area methods.

Timed Gravimetric Method The flow rate is calculated by weighing the entire content of the flow stream that was collected in a container for a fixed length of time. This is practical for small streams of less than 25 to 30 gallons per minutes (gpm) and is not well suited for continuous measurement.

Dilution Method The flow rate is measured by determining how much the flowing water dilutes an added tracer solution.

Velocity-Area Method Measuring the mean flow velocity across a cross section and multiplying it by the area at that point to calculate the flow rate.

Hydraulic Structure Method This method uses a hydraulic structure placed in the flow stream of the channel to produce flow properties that are characterized by known relationships between the water level measurement at some location and the flow rate of the stream. Therefore, the flow rate is determined by taking a single measurement of the water surface level in or near the restriction of the hydraulic structure.

Slope-hydraulic Radius-Area Method Measurement of water surface slope, cross-sectional area, and wetted perimeter over a length of uniform section channel are used to calculate the flow rate, by using a resistant equation such as the Manning formula.

The Gravitational, Dilution, and the Velocity Area methods are more commonly used for calibration purposes. The Depth-Related methods (Hydraulic Structures) are the most common. The depth-related technique measures flow rate from a measurement of the water depth, or head. Weir and flumes are the oldest and most common devices used for measuring open channel flows

Self-Check -3	Written Test

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

- Explain about measuring flows in open channel?
- What different types of measuring method?
- Which one is the most common method?

Information Sheet 4	Open channel

Introduction

An open channel is a duct in which the liquid flows with a free surface. This is in contrast with pipe flow in which the liquid completely fills the pipe and flow under pressure. The flow in a pipe takes place due to difference of pressure (pressure gradient), whereas in open channel it is due to the slope of the channel bed (i.e.; due to gravity). It may be noted that the flow in a closed conduit is not necessarily a pipe flow. It must be classified as open channel flow if the liquid has a free surface.

Information Sheet 5	Types Open channel

Introduction

Channels where flow occurs under free surface can either be

4. natural, such as rivers and
5. streams, or artificial. Artificial

channels comprise all man-made channels, including irrigation and navigation canals, spillway channels, sewers, culverts and drainage ditches. They are normally of regular cross-sectional shape and bed slope, and as such are termed prismatic channels. Their construction materials are varied, but commonly used materials include concrete, steel and earth. The surface roughness characteristics of these materials are normally well defined within engineering tolerances. In consequence, the application of hydraulic theories to flow in artificial channels will normally yield reasonably accurate results. Various terms are used to refer to channels built under different conditions. Canal: a channel built on ground, i.e excavated to the desired shape and slope with or without lining, usually having a mild slope. The lining could be made of concrete, stone masonry, cement, wood or bituminous material. Flume: a channel built (or supported) above the ground to convey fluid from one point to another. In the field flumes are made of concrete, wood, sheet metal or masonry. Laboratory flumes are usually made of wood, metal, glass or a composite of these materials. Chute: is a channel of steep slopes. If the change in elevation in the direction of flow occurs in a relatively short distance the channel is called a drop.Culvert:is a relatively short and usually buried conduit that is commonly used for drainage purposes, as in highways and embankments. Open channel prevails whenever the culvert is flowing partially full.In contrast, natural channels are normally very irregular in shape, and their materials are diverse. The surface roughness of natural channels changes with time, distance and water surface elevation. Therefore, it is more difficult to apply hydraulic theory to natural channels and obtain satisfactory results. Many applications involve man-made alterations to natural channels (e.g. river control structures and flood alleviation measures). Such applications require an understanding not only of hydraulic theory, but also of the associated disciplines of sediment transport, hydrology and river morphology. Various geometric properties of natural and artificial channels need to be determined for hydraulic purposes. In the case of artificial channels, these may all be expressed algebraically in terms of the depth (y), as isshown in Table 1.1. This is not possible for natural channels, so graphs or tables relating them to stage (h) must be used.

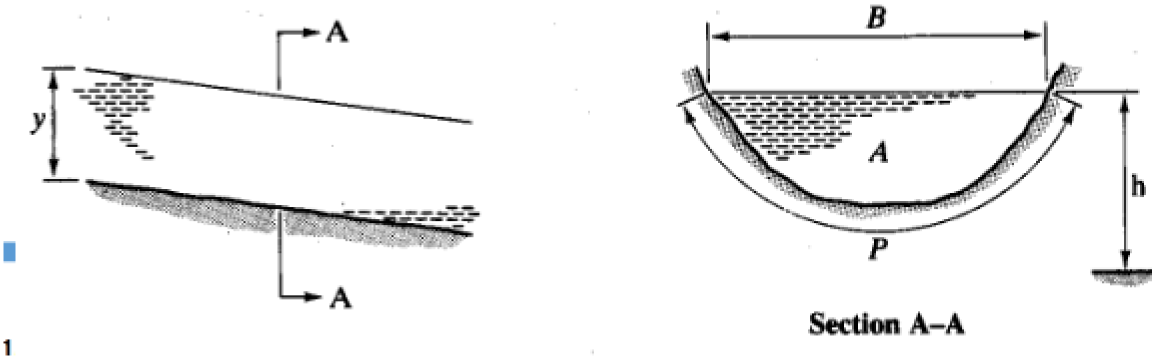


Figure 4.1 Definition sketch of geometric channel properties

Important terms in open channel flow

Depth (y)- the vertical distance of the lowest point of a channel section from the free surface;

Stage (h)- the vertical distance of the free surface from an arbitrary datum;

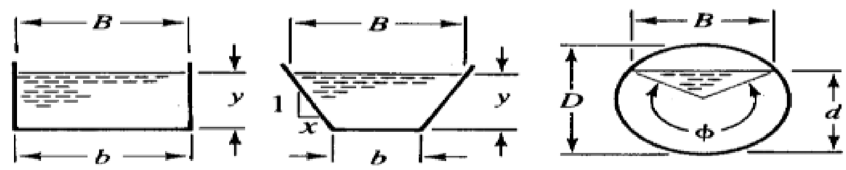
Area (A)- the cross-sectional area of flow normal to the direction of flow

Wetted perimeter (P)- the length of the wetted surface measured normal to the direction of flow;

Surface width (B)- the width of the channel section at the free surface;

Hydraulic radius (R)- the ratio of area to wetted perimeter (A / P);

Hydraulic mean depth (Dm)-t he ratio of area to surface width (A / B)



	Rectangle	Trapezoid	Circle
area, A	by	$(b+xy)y$	$\frac{1}{8}(\phi - \sin\phi)D^2$
wetted perimeter, P	$b+2y$	$b+2y\sqrt{1+x^2}$	$\frac{1}{2}\phi D$
top width, B	b	$b+2xy$	$\left(\sin\frac{\phi}{2}\right)D$
hydraulic radius, R	$\frac{by}{b+2y}$	$\frac{(b+xy)y}{b+2y\sqrt{1+x^2}}$	$\frac{1}{4}\left(1-\frac{\sin\phi}{\phi}\right)D$
hydraulic mean depth, D_m	y	$\frac{(b+xy)y}{b+2xy}$	$\frac{1}{8}\left(\frac{\phi-\sin\phi}{\sin(1/2\,\phi)}\right)D$

Table 4.1 Definition and sketches of some Geometric channel properties

Information Sheet 6	Characteristics of Open Channel

Velocity distribution in open channels

The measured velocity in an open channel will always vary across the channel section because of friction along the boundary. Neither is this velocity distribution usually axisymmetric (as it is in pipe flow) due to the existence of the free surface. It might be expected to find the maximum velocity at the free surface where the shear force is zero but this is not the case. The maximum velocity is usually found just below the surface. The explanation for this is the presence of secondary currents which are circulating from the boundaries towards the section centre and resistance at the air/water interface. These have been found in both laboratory measurements and 3d numerical simulation of turbulence. The figure below shows some typical velocity distributions across some channel cross sections. The number indicates percentage of maximum velocity

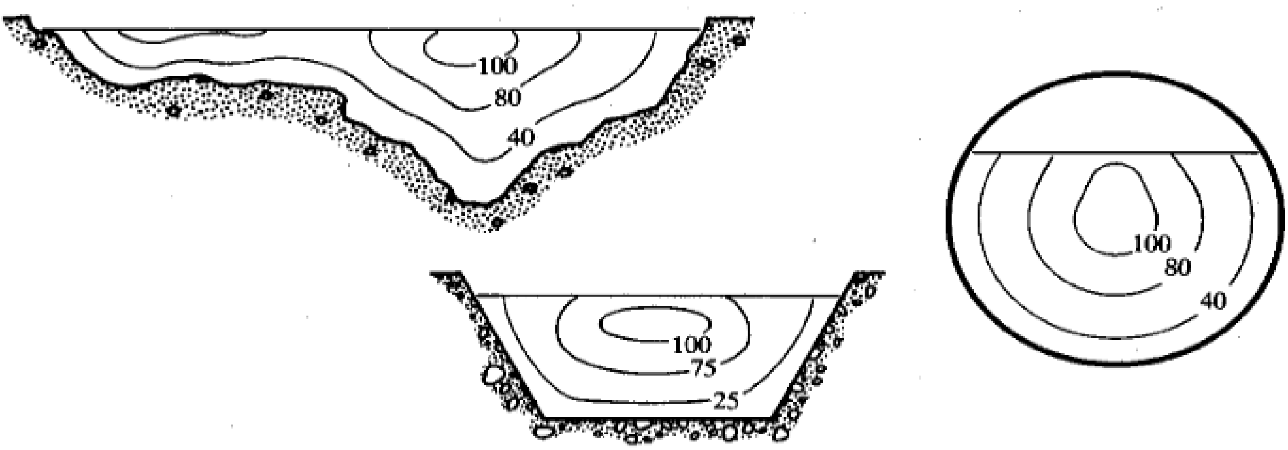


Figure 6.1 velocity distribution in open channels

Self-Check -6	Written Test

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1, Explain about open channel?

2, What are types of openchannel?

3, What is the difference b/n pipe and channel?

4, What is hydraulic jump ?

***Note:* Satisfactory rating – 30 points** **Unsatisfactory - below 25 points**

You can ask you teacher for the copy of the correct answers

Score =

Answer Sheet

Name: ___________________________ Date: ________________

Short Answer Questions

1. ____

____

2. ____

3. ____

4. ____

Instruction Sheet 4	Learning Guide 45: Calculate flows through notches and weirs

This learning guide is developed to provide you the necessary information regarding the following **content coverage** and topics:

- Notches and weirs
- Calculating flows in notches and weirs
- Characteristics of notches and weirs
- Measuring instruments and devices for notches and weirs
- Types of meters
- Limitations of the meters

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, **upon completion of this Learning Guide, you will be able to**:

- . The methods used for measuring flows in notches and weirs are identified.
- The formulae are used for calculating flows in notches and weirs.
- The applications and characteristics of notches and weirs are distinguished.
- The uses of different measuring instruments and devices used for notches and weirs are distinguished.
- The hydraulic principles which apply to different meters are assessed.
- The limitations of the meters are identified.

Learning Instructions:

2.2. Read the specific objectives of this Learning Guide.

2.3. Follow the instructions described below

2.4. Read the information written in the “Information Sheets 1- 6”. Try to understand what are being discussed.

2.5. Accomplish the “Self-checks1,2,3,4 ,5 and 6 ” in each information sheets on pages 4,11,14,19,23 and 26.

2.6. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-checks).

2.7. After You accomplish Operation sheets and LAP Tests, ensure you have a formative assessment and get a satisfactory result; then proceed to the next LG.

Information sheet -1	Notches and weirs

Introduction

We were discussing the basic concepts and also difference between Notches and Weirs in the subject of fluid mechanics in our recent posts.

Now we will go ahead to find out the various classifications of Notches and Weirs with the help of this post.

Notch

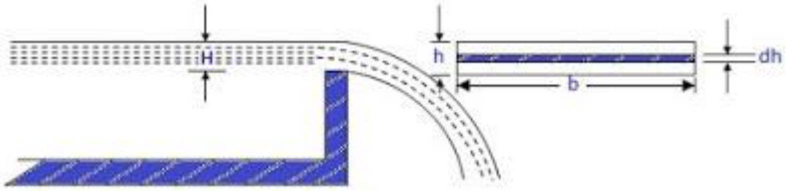
Notch is basically defined as a device which is used for determining the flow of liquid through a small channel or a tank.

Notches might be defined as the opening provided in one side of a tank or reservoir or a small channel in such a way that the liquid surface in the tank or channel is below the top edge of opening.

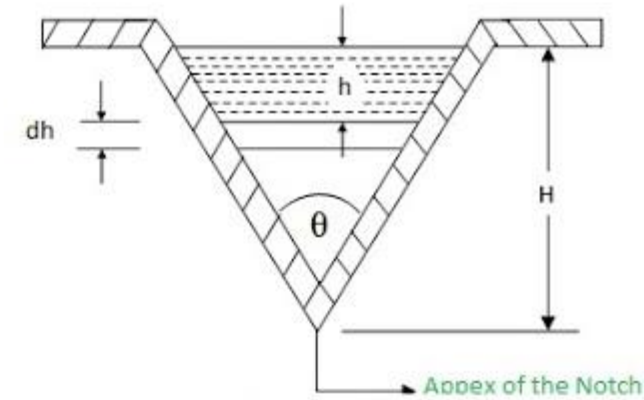
Classification of notches

There are following types of notches on the basis of shape of opening and these are as mentioned here

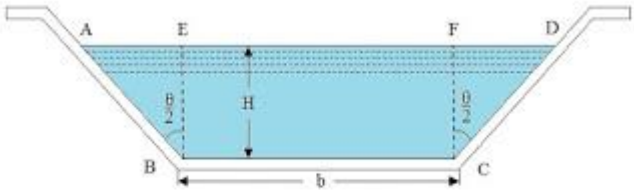
Rectangular Notch



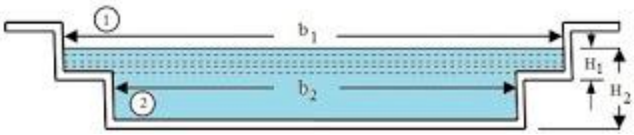
Triangular Notch or V- notch



Trapezoidal Notch



Stepped Notch



There are following types of notches on the basis of the effect of sides on the nappe and these are as mentioned here

1. Notch with end contraction
2. Notch without end contraction or suppressed notch

Weir

A weir will be basically a concrete or masonary structure which will be located in an open channel over which flow will take place.

We can also define as the structure constructed across the river or large canal for storing water on upstream side.

Weir will be usually in the form of vertical wall, with a sharp edge at the top, running all the way across the open channel.

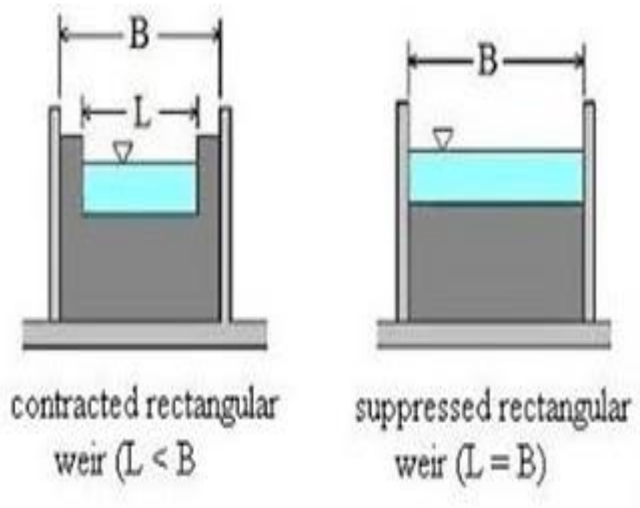
Classification of weirs

There are following types of weirs on the basis of shape of opening and these are as mentioned here

1. Rectangular weir
2. Triangular weir
3. Trapezoidal or Cippoletti weir

There are following types of weirs on the basis of shape of the crest and these are as mentioned here

1. Sharp crested weir
2. Broad crested weir
3. Narrow crested weir
4. Ogee shaped weir



There are following types of weirs on the basis of the effect of sides on the emerging nappe and these are as mentioned here

1. Weirs with end contraction
2. Weirs without end contraction or suppressed notch

Self-Check -1	Written Test

Direction : answer the following questions

1. What is Notch?
2. What is weir?
3. What is the different between notches and weirs?
4. Write classification of weirs?
5. Write classification of notches?

Information sheet-2	formulae are used for calculating flows in notches and weirs

Introduction

A notch means an opening provided in the side of a tank, such that the opening extends even above the free surface of the liquid in the tank. It is in a way, a large orifice having no upper edge. A notch is generally meant to measure the flow of water from a tank. A weir is also a notch but it is made on a large scale. The weir is a notch cut in a dam to discharge the surplus quantity of water.

Water flows over a notch or weir while water passes through an orifice. While the stream of water discharged by an orifice is called a jet, the sheet of water discharged by a notch or weir is called a nappe or vein. The upper surface of the notch or weir over which the water flows is called the Crest or Sill

A notch or a weir is a convenient device for the measurement of discharge in an open channel. A notch or a weir is an obstruction provided in a channel that causes the water to rise behind it so that the water is made to flow through it or over it. The rate of flow can be determined by measuring the height of the upstream water level.

Basically there is no difference between a notch and a weir, except that a notch is of small size while a weir is of large size. A notch is usually made of metal plate whereas a weir is made of masonry or concrete.

1.1 Calculating Notches

Notch is basically defined as a device which is used for determining the flow of liquid through a small channel or a tank. Notches might be defined as the opening provided in one side of a tank or reservoir or a small channel in such a way that the liquid surface in the tank or channel is below the top edge of opening.

Classification of Notches

1. The Rectangular Notch:

Consider a rectangular notch shown in Fig. 9.2.

Let l = Length of the notch

H = Head of water over the crest of the notch.

Consider an elemental horizontal strip of water of length l and thickness dh, at a depth h below the free surface of water.

Theoretical velocity of water flowing through the elemental strip $= \sqrt{2gh}$

$\therefore$ Theoretical discharge through the elemental strip $= l\,dh\sqrt{2gh}$

$$\therefore \text{Total theoretical discharge} = Q = l\sqrt{2g}\int_0^H h^{1/2}\,dh = \frac{2}{3} l\sqrt{2g}H^{3/2}$$

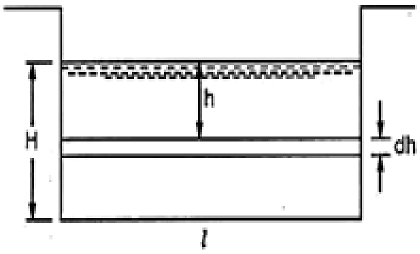


Fig. 9.2.

$$\text{Actual discharge, } = q = \frac{2}{3}C_d l\sqrt{2g}H^{3/2}$$

Where $= C_d$ = Coefficient of discharge.

▪ ***The Triangular Notch or V-Notch***

Fig. 9.3 shown a triangular notch.

Let H = head of water over the apex

θ = Angle of the notch

Width of the notch at any depth h

$$=2(H-h)\tan\frac{\theta}{2}$$

Consider an elemental horizontal strip of the opening at depth h and having a height dh. The theoretical velocity of flow through the strip $=\sqrt{2gh}$

$\therefore$ Theoretical discharge through the strip

$$=2(H-h)\tan\frac{\theta}{2}dh\sqrt{2gh}$$

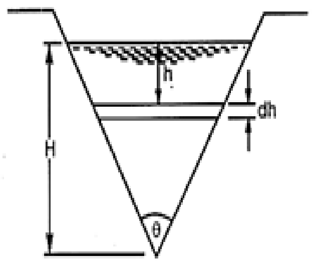


Fig. 9.3.

$$\text{Total discharge } = Q = \int_0^H 2\sqrt{2g}\tan\frac{\theta}{2}(H-h)h^{1/2}dh$$

$$= 2\sqrt{2g}\tan\frac{\theta}{2}\left[H\frac{2}{3}H^{3/2}-\frac{2}{5}H^{5/2}\right]=\frac{8}{15}\sqrt{2g}\tan\frac{\theta}{2}H^{5/2}$$

$$\text{Actual discharge } = q = \frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}H^{5/2}$$

where C_d = Coefficient of discharge

The vertex angle for a triangular notch may be from 25° to 90°. A vertex angle of 90° is commonly adopted. The coefficient of discharge is found to depend on the vertex angle. At lower heads and lower vertex angles the values of C_d are found to be higher. This may be due to a lesser degree of contraction of the nappe.

$$\text{For a } 90° \text{ notch } \tan\frac{\theta}{2} = 1$$

$$\text{and the discharge } = q = \frac{8}{15}C_d\sqrt{2g}H^{5/2}$$

Taking $C_d = 0.6$, we have

$$\frac{8}{15}C_d\sqrt{2g} = \frac{8}{15}\times 0.6\sqrt{2\times 9.81} = 1.47$$

and accordingly $q = 1.417\ H^{5/2}$.

3. The Trapezoidal Notch:

Consider a trapezoidal notch whose crest length is l and the sides are at θ with the vertical. Let H be the head of water over the crest

In this case the notch may be taken to consist of a rectangular notch of length l and a triangular notch subtending an angle 2θ.

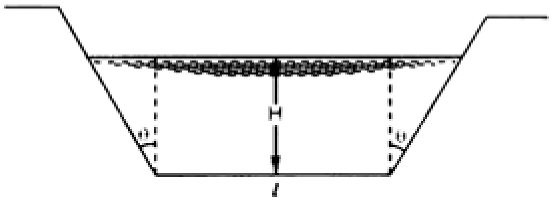


Fig. 9.4. The trapezoidal notch.

Total discharge = q = discharge over the rectangular notch + discharge over the triangular notch.

$$q = \frac{2}{3} C_d l \sqrt{2g}\, H^{3/2} + \frac{8}{15} C_d \sqrt{2g} \tan\theta \cdot H^{5/2}$$

$$= C_d \sqrt{2g}\, H^{3/2} \left(\frac{2}{3} l + \frac{8}{15} \tan\theta . H \right)$$

Advantages of a Triangular Notch over Rectangular Notch:

A triangular notch has certain advantages over the rectangular notch when used as a gauging device in a hydraulic laboratory.

The advantages are:

(i) The coefficient of discharge for a triangular notch is practically independent of the head. This is because, for all heads the ratio of the head to the wetted length or crest is constant. But in a rectangular notch the ratio of the head to the wetted length crest is not constant. Hence for a rectangular notch the coefficient of discharge is not actually a constant but is a function of the head over the notch.

(ii) When the discharge rate is small a triangular notch provides a greater head than the rectangular notch. Hence head measurement can be done more accurately over the triangular notch than over the rectangular notch.

(iii) When the discharge rate is small, there are chances of a clinging nappe to be formed when a rectangular notch is used. But for the same discharge over the triangular notch the head will be greater and the clinging nappe will be avoided.

(iv) When a triangular notch is provided, there will be no need for any special arrangement for ventilating the nappe.

1.2 Calculating Weir

A weir will be basically a concrete or masonary structure which will be located in an open channel over which flow will take place. We can also define as the structure constructed across the river or large canal for storing water on upstream side. Weir will be usually in the form of vertical wall, with a sharp edge at the top, running all the way across the open channel. Weirs are common and simple methods of measuring the flow of water in open channels. At its simplest, a weir is no more than an obstruction placed in a channel over which water flows (unlike flumes where the water flows through the structure). Often this flow is over a specially shaped notch or opening set above the floor of the channel.

Properly size, built, and maintained, a weir can provide inexpensive and accurate flow measurements.

Open channel flow offers a variety of weirs, including: fixed weir plates, portable weir sets, weir boxes, weir channels, and weir manholes.

Classification of Weirs:

1. Proportional Weir:

This is a weir whose shape is so designed that the discharge over the weir is proportional to the head of water over the crest. The crest of the weir is horizontal.

Consider the weir shown in Fig. 9.13, whose equation with the middle point of the crest as origin is $x^2 = K/y$ where K is a constant. Let H be the head of water over the weir. Consider a horizontal strip of the flow section of width 2x at a height y above the crest. Let dy be the thickness of the strip. Discharge through the elemental horizontal strip.

$$= 2KC_d\sqrt{2g}H\left(\frac{\pi}{2}\right)$$

$$\therefore \quad Q = (KC_d\sqrt{2g}\pi)H = \text{constant} \times H$$

The above equation shows that the discharge is proportional to the head of water over the crest of he weir. The proportional weir is not theoretically possible because as per the equation $x^2 = \frac{K}{y}2x$ the width $2x$ becomes infinite at $y = 0$ *i.e.*, the crest width should be infinity, which is not practicable.

To overcome this difficulty, the shape of the weir was modified by *Sutro* by providing a finite width l at the crest. The equatioin of the sides of the weir was modified. He also introduced *a* rectangular zone of depth *a*. Sutro's equation for the side width is,

$$\frac{2x}{l} = \left(1 - \frac{2}{\pi}\tan^{-1}\sqrt{\frac{y}{a}}\right)$$

The discharge is given by

$$Q = \left(C_d l\sqrt{2ga}\right)\left(H - \frac{a}{3}\right)$$

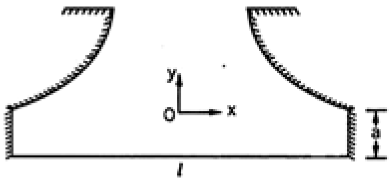


Fig. 9.14. Sutro weir.

2. The Cippoletti Weir:

We know for a rectangular weir with the two end contractions, the discharge is given by –

3. Submerged Weir:

Fig. 9.19 shows a submerged weir. In this case the water level on the downstream side also is above the crest of the weir. Let H_1 and H_2 be the heights of the upstream and downstream water levels above the crest weir. The total discharge Q consists of components Q_1 and Q_2.

4. Anicut or Raised Weir or Barrage:

An anicut is a masonry dam provided across a river for the purpose of raising the water level on the upstream side to a sufficient extent in the dry season, so that the water can be carried by gravitation to places where it otherwise could not reach.

Time of Emptying a Reservoir by a Rectangular Weir:

Let a reservoir of plan area A be provided with a rectangular weir of length l. Let it be required to find the time taken for the head of water over the weir to fall from a value H_1 to a value H_2.

Let at any instant the head of water over the weir be h. Let the fall in water level be dh in a small interval of time dt

$\therefore$ Quantity discharged in the interval of time dt

$$= dq = AdH = \frac{2}{3} C_d l \sqrt{2g} h^{3/2} dt$$

$$\therefore \quad dt = \frac{A}{\frac{2}{3} C_d l \sqrt{2g}} h^{-3/2} dh$$

$\therefore$ Interval of time required to change the head form H_1 to H_2 is obtained by integrating the above expression from the lower value of h to upper value of h

$$\therefore \quad T = \frac{A}{\frac{2}{3} C_d l \sqrt{2g}} \int_{H_1}^{H_2} h^{-3/2} dh = \frac{2A}{\frac{2}{3} C_d l \sqrt{2g}} \left(\frac{1}{\sqrt{H_2}} - \frac{1}{\sqrt{H_1}} \right)$$

Time of Emptying a Reservoir by a Triangular Notch:

Let a reservoir of plan area A be provided with a triangular notch of angle θ. Let it be required to find the time taken for the head of water over the notch to fall from a value H_1 to a value H_2. Let at any instant the head of water over the notch be h. Let the fall in water level be dh in a small interval of time dt.

$\therefore$ Quantity discharged in the interval of time dt

$$= dq = Adh = \frac{8}{15} C_d \sqrt{2g} \tan\frac{\theta}{2} h^{5/2} dt$$

$$\therefore \quad dt = \frac{A}{\frac{8}{15} C_d \sqrt{2g} \tan\frac{\theta}{2}} h^{-5/2} dh$$

$\therefore$ Interval of time required to change the head from H_1 to H_2 is obtained by integrating the above expression from the lower value of h to the upper value of h

$$\therefore \quad T = \frac{A}{\frac{8}{15} C_d \sqrt{2g} \tan\frac{\theta}{2}} \int_{H_2}^{H_1} h^{-5/2} dh$$

$$= \frac{\frac{2}{3} A}{\frac{8}{15} C_d \sqrt{2g} \tan\frac{\theta}{2}} \left(\frac{1}{H_2^{3/2}} - \frac{1}{H_1^{3/2}} \right)$$

5. Broad Crested Weir:

This is a weir having a very broad sill so that the flow of water over the sill may be compared to the flow of water in a channel. Consider the broad crested weir shown in Fig. 9.25.

Let H be the head of water over the weir. Let l be the length of the weir. As the water flows over the weir, it reaches a uniform depth of flow h, over the crest. Let v be the uniform velocity of flow over the weir.

Applying Bernoulli's equation to the still water surface on the upstream side and the running liquid surface at the outlet.

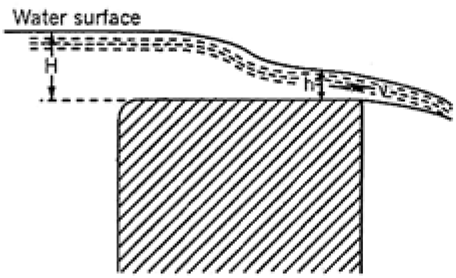


Fig. 9.25. Broad crested weir.

$$H = h + \frac{v^2}{2g} \text{ ignoring losses}$$

$$v = \sqrt{2g(H-h)}$$

$$\text{Discharge per second} = Q = \text{Area} \times \text{Velocity} = lh\sqrt{2g(H-h)}$$

This is the theoretical discharge.

$$\therefore \quad \text{Actual discharge} = q = C_d lh\sqrt{2g(H-h)}$$

where C_d is a coefficient of discharge.

Condition for maximum discharge

We know $\quad q = C_d lh\sqrt{2g(H-h)} = C_d l\sqrt{2g}\sqrt{Hh^2 - h^3}$

For q to be maximum, the quantity $(Hh^2 - h^3)$ should be maximum.

$$\frac{d}{dh}(Hh^2 - h^3) = 0$$

$$2Hh - 3h^2 = 0$$

$$\therefore \quad h = \frac{2}{3}H$$

$$\therefore \quad q_{max} = C_d l\frac{2}{3}H\sqrt{2g(H - \frac{2}{3}H)} = \frac{2}{3}\sqrt{\frac{2g}{3}}C_d lH^{3/2}$$

$$q_{max} = 1.705\ C_d lH^{3/2}$$

6. Ogee Weir:

We know, in the case of a sharp crested weir the nappe as it leaves the crest springs or rises slightly at the lower surface. In this way it reaches a maximum rise of 0.115 H' above the crest and then falls. (H' is the head over sharp crest). Suppose the space below the bottom surface of the nappe be filled with masonry or concrete.

The consequent weir formed is shown in Fig. 9.26. Such a weir is called an ogee weir. Thus in an ogee weir, the solid boundary of the weir exactly coincides with the bottom surface of the nappe of the sharp crested weir under the designed head.

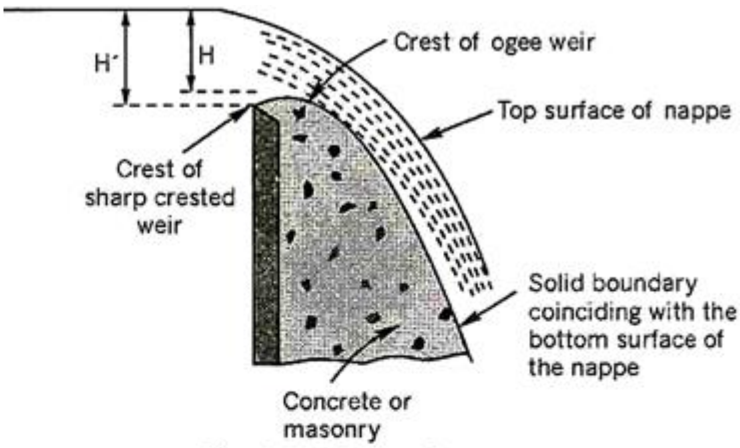


Fig. 9.26. Ogee wier.

If H' is the head above the sharp crest then by Francis formula the discharge is given by, $q = 1.84\ lH'^{3/2}$. Alternatively if the head H above the spillway be considered then the discharge is given by, $q = 2.20\ lH^{3/2}$.

7. Separating Weir:

A separating weir is an arrangement provided in the case of town water supply, where it may become necessary to divert the discoloured flood water from the supply channel. When the discharge is moderate, the water drops over the lip C into a culvert D which communicates with the supply channel.

But during floods, the velocity of flow will be greater due to greater depth and this causes the water to leap across the opening into the waste channel.

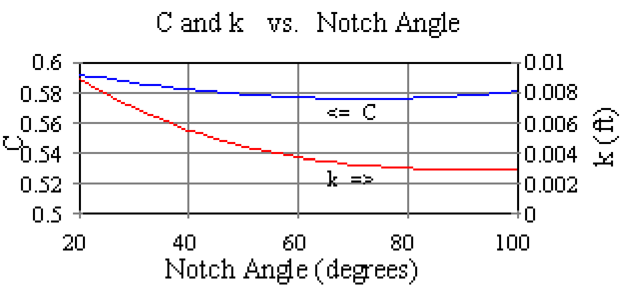
Self-Check -2	Written Test

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page

- The graph shown is from our fits. If you compare it to the graphs shown in the references, it looks nearly identical which implies that our fits are very good.

$$Q = 4.28\, C \tan\left(\frac{\theta}{2}\right) (h + k)^{5/2}$$

where Q = Discharge (cfs)
C = Discharge Coefficient
θ = Notch Angle
h = Head (ft)
k=Head Correction Factor (ft)



C = 0.607165052 - 0.000874466963 θ + 6.10393334x10^{-6} θ^2
k (ft.) = 0.0144902648 - 0.00033955535 θ + 3.29819003x10^{-6} θ^2 - 1.06215442x10^{-8} θ^3
where θ is the notch angle in degrees?

2. The flow rate measurement in a rectangular weir is based on the Bernoulli Equation principles and can be expressed as:

$$q = 2/3\, c_d\, b\, (2\, g)^{1/2}\, h^{3/2} \qquad (1)$$

where

q = flow rate (m^3/s)

h = elevation head on the weir (m)

b = width of the weir (m)

g = 9.81 (m/s^2) - gravity

c_d = discharge constant for the weir - must be determined

c_d must be determined by analysis and calibration tests. For standard weirs - c_d - is well defined or constant for measuring within specified head ranges.

The lowest elevation *(h = 0)* of the overflow opening of the sharp-crested weirs or the control channel of broad-

Information Sheet 3	characteristics of notches and weirs

The characteristics of weirs

They have different in a shape also the applications' for example

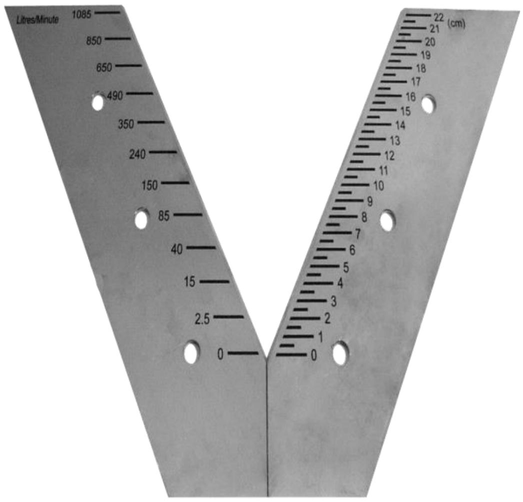
Sharp-Crested Weirs

Sharp-crested weirs are typically constructed by placing a thin, rust resistant metal plate, with a notch in the top of it, perpendicular to the flow of water (concrete and timber can also be used to construct). Water will flow through the notch and the depth of water that flows through it will correlate to the discharge in the channel (Cruise). There are three main types of sharp-**crested weirs**: rectangular, triangular, and trapezoidal

Rectangular Weirs -

Rectangular weirs are typically used to control the elevation of water up and downstream of

the weir and they usually have higher discharge values associated with them. There are two main types of rectangular weirs. The first type is a suppressed weir, where the crest stretches across the whole width of the channel



Common Weir Terminology:

Notch – the opening through which water flows

Crest – the edge which water flows over

Nape – the sheet of water that flows over the weir

Length – the “width” of the weir notch

Triangular (V-Notch) Weirs –

In cases with small discharge, triangular or v-notch weirs are typically used. Because of the

lower flow rate, these weirs are very good for measuring the discharge in an open channel. The smaller flow area makes for a larger upstream head that is also easier to measure. This type of weir generally has the highest accuracy in measuring flow rates

Some advantages of using broad-crested weirs for flow measurement and regulation:

• Cost effective installation due to ease of design and construction

• Relatively small head loss across the structure

• Sturdy and capable of measuring discharge in small to medium channels

• Theoretical calibration possible based on post-construction dimensions

• Capable of passing floating debris

Some disadvantages of using broad-crested weirs for flow measurement and regulation:

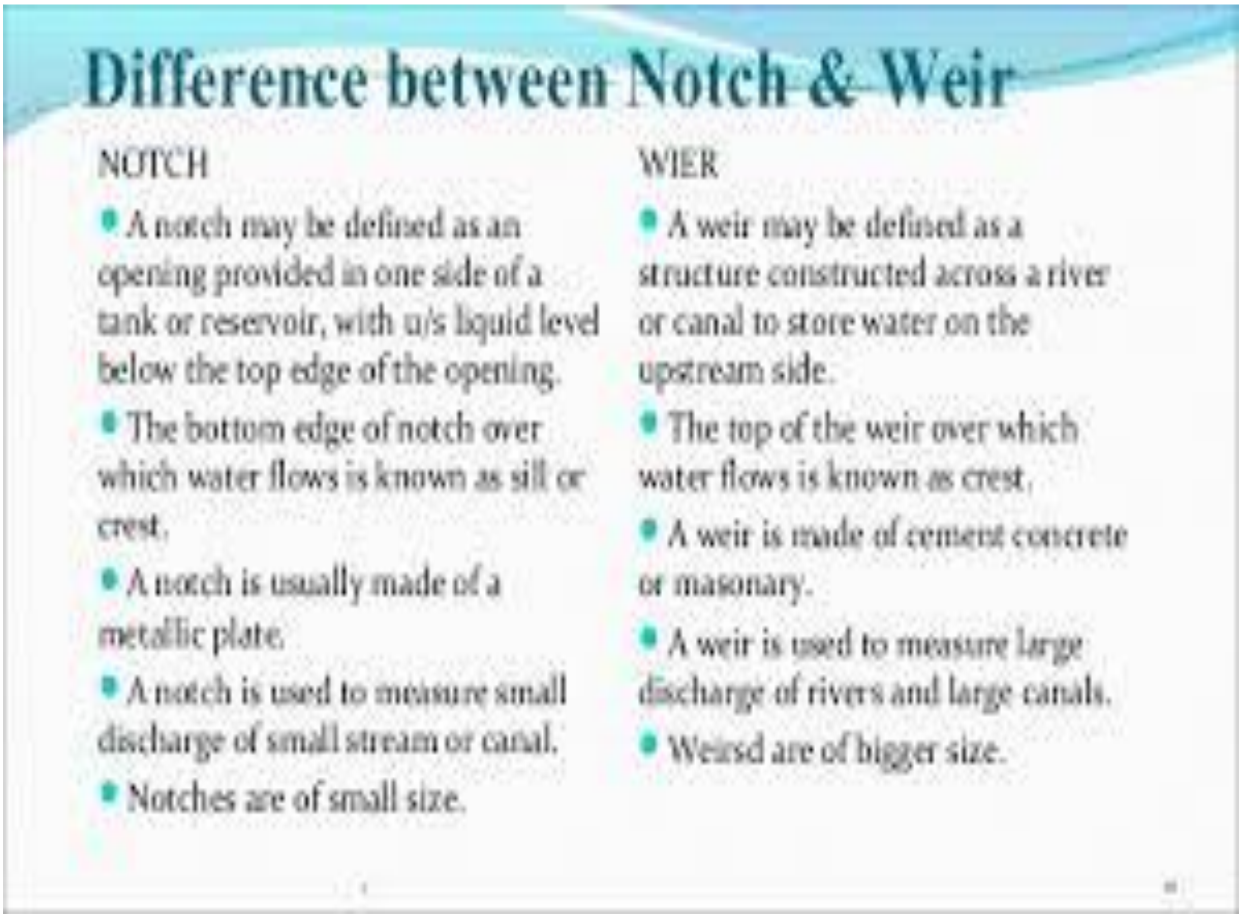
• May interfere with fish passage and disrupt ecological equilibrium

• Sediment deposition occurs on the upstream side of the structure, leading to lower sediment flow downstream and higher water levels upstream

• The channel immediately upstream of the weir is prone to sediment deposition which in turn can compromise the accuracy of the rating curve

• Head loss occurs across the weir (especially when there is a hydraulic jump)

The characteristics of notches in a velocity approach



It is defined as the velocity with which the flow approaches/reaches the notch/weir before it flows past it. The velocity of approach for any horizontal element across the notch depends only on its depth below the free surface. In most of the cases such as flow over a notch/weir in the side of the reservoir, the velocity of approach may be neglected. But, for the notch/weir placed at the end of the narrow channel, the velocity of approach to the weir will be substantial and the head producing the flow will be increased by the kinetic energy of the approaching liquid. Thus, if V_a is the velocity of approach, then the additional head H_a due to velocity of approach, acts on the water flowing over the notch or weir. So, the initial and final height of water over the notch/weir will be $(H + H_a)$ and H_a respectively. It may be determined by finding the discharge over the notch/weir neglecting the velocity of approach i.e.

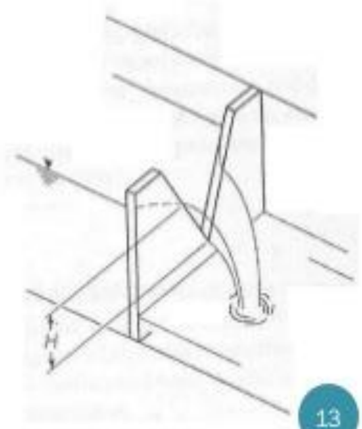
$$V_a = \frac{Q}{A} \qquad (1)$$

where Q Q is the discharge over the notch/weir and A A is the cross-sectional area of channel on the upstream side of the weir/notch. V Additional head corresponding to the velocity of approach will be

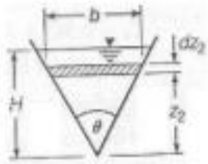
V-notch weir

- A V-notch weir is a sharp-crested weir that has a V-shaped opening instead of a rectangular-shaped opening. These weirs, also called triangular weirs, are typically used instead of rectangular weirs under low-flow conditions, where rectangular weirs tend to be less accurate.
- V-notch weirs are usually limited to flows of 0.28 m^3/s (10 cfs) or less. The flow rate, Q, over a V-notch weir is therefore given by

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



- C_d coefficient of discharge
- g gravitational acceleration (m/s^2)
- θ angle of V shape (°)
- H water elevation over the crest of weir (m)



13

Generally the application of wiers and notches are

The rectangular weir (notch) is a common device used to regulate and measure discharge in irrigation projects. The current research was based mainly on laboratory experiments studying the hydraulic characteristics of rectangular notches. Four rectangular notches were used in this research in different models. Notches for all models were designed with the same shape, arrangement, and width (4 cm), but differed in height, with examples at 6, 8, 10, and 12 cm. The main objective of this research was to study the influence of rectangular notch dimensions and upstream water depth on discharge coefficients. The results obtained from this research indicate that the relationship between the discharge coefficient and the upstream water depth is a power function. The values of the discharge coefficient increase with increases in the values of the upstream water depth. The relationship between the discharge coefficient and the Reynolds number is also a power function, and an increase in the Reynolds number leads to a decreased discharge coefficient. In addition, when the value of the Reynolds number is high (turbulent flow), the values of the discharge coefficient converge to an approximately constant value. The flow in all runs was subcritical and the relationship between discharge coefficient and Froude number was also found to be a power function. An increase in Froude number thus leads to a decrease in discharge coefficient. The slope of the discharge coefficient-Froude number curve values gradually decreases until the value the discharge coefficient reaches an approximately constant value. A dimensional analysis technique was used to estimate the values of the discharge coefficient for various rectangular notch dimensions, and an empirical equation for discharge coefficient estimating was derived using regression procedure. This equation has a coefficient of determination R^2 of 0.955.

Uniform flow

The equations that are used in uniform flow calculations are,

a) Continuity equation,

$$Q = VA$$

b) Manning velocity equation,

$$V = \frac{1}{n} R^{2/3} S_0^{1/2}$$

$$Q = AV = A \frac{1}{n} R^{2/3} S_0^{1/2}$$

The water depth in a channel for a given discharge Q and n= Manning coefficient, S0= Channel slope, B= Channel width, is called as Uniform Water Depth.

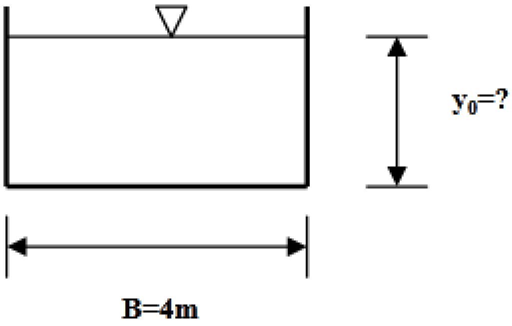
The basic variables in uniform flow problems can be the discharge Q, velocity of flow V, normal depth y0, roughness coefficient n, channel slope S0 and the geometric elements (e.g. B and side slope m for a trapezoidal channel). There can be many other derived variables accompanied by corresponding relationships. From among the above, the following five types of basic problems are recognized.

Problem Type	Given	Required
1	y_0, n, S_0, Geometric elements	Q and V
2	Q, y_0, n, Geometric elements	S_0
3	Q, y_0, S_0, Geometric elements	n
4	Q, n, S_0, Geometric elements	y_0
5	Q, y_0, n, S_0, Geometry	Geometric elements

Problems of the types 1, 2 and 3 normally have explicit solutions and hence do not represent any difficulty in their calculations. Problems of the types 4 and 5 usually do not have explicit solutions an as such may involve trial-and-error solution procedures.

Example 4.1: Calculate the uniform water depth of an open channel flow to convey Q=10 m3/sec discharge with manning coefficient n=0.014, channel slope S0=0.0004, and channel width B=4 m.

a)Rectangular cross-section



$$A = By_0 = 4y_0$$
$$P = B + 2y_0$$
$$R = \frac{A}{P} = \frac{4y_0}{4+2y_0}$$

$$Q = AV = A\frac{1}{n}R^{2/3}S_0^{1/2}$$

$$10 = 4 \times y_0 \times \frac{1}{0.014} \times \left(\frac{4y_0}{4+2y_0}\right)^{2/3} \times 0.0004^{1/2}$$

$$y_0 \times \left(\frac{y_0}{4+2y_0}\right)^{2/3} = \frac{10 \times 0.014}{4^{5/3} \times 0.02} = 0.694$$

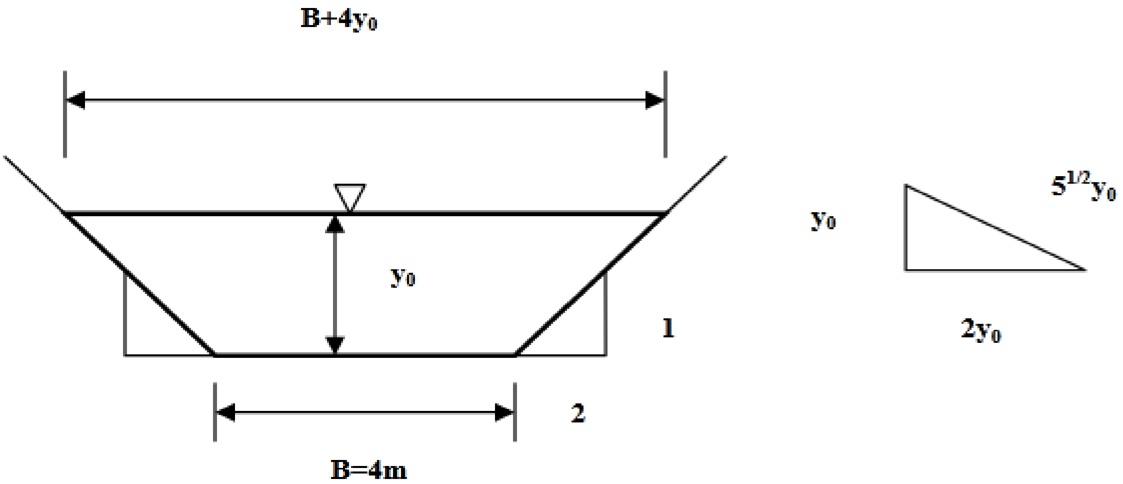
$$X = \frac{y_0^{5/3}}{(4+2y_0)^{2/3}}$$

$$y_0 = 2m \rightarrow X = 0.794 \neq 0.694$$
$$y_0 = 1.90m \rightarrow X = 0.741 \neq 0.694$$
$$y_0 = 1.80m \rightarrow X = 0.689 \neq 0.694$$
$$y_0 = 1.81m \rightarrow X = 0.694 \cong 0.694$$

The uniform water depth for this rectangular channel is y_0 = 1.81 m.

There is no implicit solution for calculation of water depths. Trial and error must be used in calculations.

b)Trapezoidal Cross-Section



$$A = \frac{B + B + 4y_0}{2} y_0 = (B + 2y_0)y_0 = (4 + 2y_0)y_0$$

$$P = B + 2\times\sqrt{5}\times y_0 = 4 + 4.472y_0$$

$$R = \frac{A}{P} = \frac{(4 + 2y_0)y_0}{4 + 4.472y_0}$$

Trial and error method will be used to find the uniform water depth

$$Q = AV = A\frac{1}{n}R^{2/3}S_o^{1/2}$$

$$10 = (4y_0 + 2y_o^2)\times\frac{1}{0.014}\times\left(\frac{4y_0 + 2y_0^2}{4 + 4.472y_0}\right)^{2/3}\times 0.0004^{1/2}$$

$$\frac{10\times 0.014}{0.02} = 7$$

$$X = \frac{(4y_0 + 2y_0^2)^{5/3}}{(4 + 4.472y_0)^{2/3}}$$

$$y_0 = 1m \rightarrow X = 4.95 \neq 7$$

$$y_0 = 1.10m \rightarrow X = 5.70 \neq 7$$

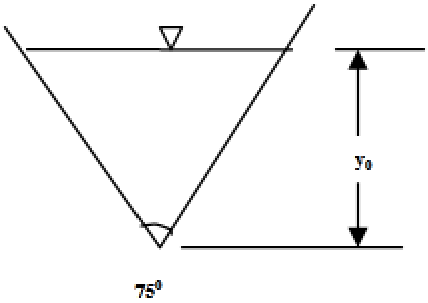
$$y_0 = 1.20m \rightarrow X = 6.73 \neq 7$$

$$y_0 = 1.22m \rightarrow X = 6.94 \neq 7$$

$$y_0 = 1.23m \rightarrow X = 7.05 \cong 7$$

The uniform water depth for this trapezoidal channel is y_0 =1.23 m.

Example 4.2: A triangular channel with an apex angle of 75^0 carries a flow of 1.20 m^3/sec at a depth of 0.80 m. If the bed slope is S_0 = 0.009, find the roughness coefficient n of the channel.



Solution:

y_0 = Normal depth = 0.80 m

Referring to Figure,

Area $$A = \frac{1}{2} \times 0.80 \times 2 \times 0.80 \times \tan\left(\frac{75}{2}\right) = 0.491 m^2$$

Wetted perimeter $$P = 2 \times 0.80 \times \sec 37.5^0 = 2.02 m$$

$$R = \frac{A}{P} = \frac{0.491}{2.02} = 0.243 m$$

$$n = \frac{A R^{2/3} S_0^{1/2}}{Q} = \frac{0.491 \times 0.243^{2/3} \times 0.009^{0.5}}{1.20} = 0.0151$$

Self-Check -4	Written Test

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

2. Calculate the uniform water depth of an open channel flow to convey Q=10 m3/sec discharge with manning coefficient n=0.014, channel slope S0=0.0004, and channel width B=4 m.
3. A triangular channel with an apex angle of 750 carries a flow of 1.20 m3/sec at a depth of 0.80 m. If the bed slope is S0 = 0.009, find the roughness coefficient n of the channel.

Information sheet -5	Types of meters

CURRENT METER

A current meter is oceanographic device for flow measurement by mechanical (rotor current meter), tilt (Tilt Current Meter), acoustical (ADCP) or electrical means.

MEASUREMENT PRINCIPLES

a. Mechanical current meters are mostly based on counting the rotations of a propeller and are thusrotor current meters. A mid-20th-century realization is theEkman current meterwhich drops balls into a container to count the number of rotations.

b.Acoustic There are two basic types of acoustic current meters: Doppler and Travel Time. Both methods use a ceramic transducer to emit a sound into the water.Doppler instruments are more common. An instrument of this type is theAcoustic Doppler Current Profiler(ADCP)which measures the water current velocities over a depth range using theDoppler effectofsound wavesscattered back from particles within the water column. The ADCPs use the traveling time of the sound to determine the position of the moving particles. Single-point devices use again the Doppler shift, but ignoring the traveling times. Such a single point Doppler Current Sensor (DCS) has a typical velocity range of 0 to 300cm/s.Travel time instruments determine water velocity by at least two acoustic signals, one up stream and one down stream.

c. Electromagnetic Induction This novel approach is for instance employed in the Florida Strait where electromagnetic induction in submerged telephone cableis used to estimate the through-flow through the gatewayand the complete setup can be seen asone huge current meter. it is possible to evaluate the variability of the averaged horizontal flow by measuring the induced electric currents. The method has a minor vertical weighting effect due to small conductivity changes at different depths.

d.Tilt. current meters operate under the drag-tilt principle. They consist of a sub-surface buoy that is anchored to the sea floor with a flexible line or tether. The float tilts as a function of its shape, buoyancy and the water velocity. Once the characteristics of a given buoy are known, the velocity can be determined by measuring the angle of the buoy.A Tilt Current Meter is typically deployed on the bottom with an anchor but may be deployed on lobster traps or other convenient anchors of opportunity.

VENTURIMETER

Venturimeter is a device used for measuring the rate of flow of a fluid flowing through a pipe. It consists of three parts:• A short converging part

Throat Diverging partLet d1 = diameter at the inlet (section 1)p1 = pressure at section 1v1 = velocity at section 1A1= area at section1 d2, p2, v2, A2 are the corresponding values at the throat (section 2) Applying Bernoulli’s equations at sections 1 and 2, we get

ORIFICE METER

An orifice meter is a conduit and a restriction to create a pressure drop. An hour glass is a form of orifice. A nozzle, venturi or thin sharp edged orifice can be used as the flow restriction. In order to use any of these devices for measurement it is necessary to empirically calibrate them.

That is, pass a known volume through the meter and note the reading in order to provide a standard for measuring other quantities. Due to the ease of duplicating and the simple construction, the thin sharp edged orifice has been adopted as a standard and extensive calibration work has been done so that it is widely accepted as a standard means of measuring fluids.

Instruction sheet -5	Learning guide 46: **Calculate proportion for economic section**

This learning guide is developed to provide you the necessary information regarding the following content coverage and topics –

- Calculating proportions of rectangular, trapezoidal and circular channels for maximum discharge
- Differentiate a partial flow chart is used to identify the depth of flow for maximum discharge and maximum velocity

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, upon completion of this Learning Guide, you will be able to –

- Understand The proportions of rectangular, trapezoidal and circular channels are calculated for maximum discharge.
- Differentiate A partial flow chart is used to identify the depth of flow for maximum discharge and maximum velocity

Learning Instructions

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below
3. Read the information written in the “Information Sheets 1- 2”. Try to understand what are being discussed.
4. Accomplish the “Self-checks1and 2 ” in each information sheets on
5. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-checks).
6. After You accomplish Operation sheets and LAP Tests, ensure you have a formative assessment and get a satisfactory result; then proceed to the next LG.

Information sheet -1	Calculating proportions of rectangular, trapezoidal and circular channels for maximum discharge

Introduction to Most Economical Section of Channels

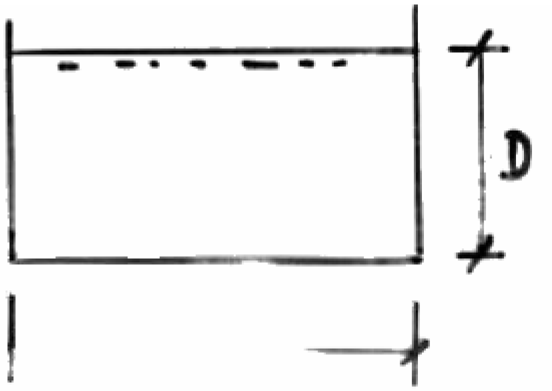
A section of a channel is said to be most economical when the cost of construction of the channel is minimum. But the cost of construction of a channel depends on excavation and the lining. To keep the cost down or minimum, the wetted perimeter, for a given discharge, should be minimum. This condition is utilized for determining the dimensions of economical sections of different forms of channels. Most economical section is also called the best section or most efficient section as the discharge, passing through a most economical section of channel for a given cross-sectional area A, slope of the bed S_0 and a resistance coefficient, is maximum. But the discharge

$$Q = AV = AC\sqrt{R_h\, S} = AC\sqrt{\frac{A}{P}S} = const.*\frac{1}{\sqrt{P}}$$

Hence the discharge Q will be maximum when the wetted perimeter P is minimum

1.1 Most Economical Rectangular Channel

Consider a rectangular section of channel as shown.



Let B = width of channel, D = depth of flow.

$\therefore$ area of flow, $A = B \, x \, D$, (7.4a)

wetted perimeter, $P = 2D + B$, (7.4b)

from Eq. (7.4a) we have $B = \frac{A}{D}$, which if substituted in (7.4b), we get

$$P = 2D + \frac{A}{D} \quad (7.4c)$$

For most economical cross section, P should be minimum for a given area

$$\frac{dP}{dD} = 0, \quad \text{so} \quad \frac{dP}{dD} = 2 - \frac{A}{D^2} = 0 \quad \Rightarrow \quad 2 = \frac{A}{D^2} = \frac{BD}{D^2}$$

$$\Rightarrow \quad 2 = \frac{B}{D} \quad \text{and hence}$$

$$D = \frac{B}{2} \quad (7.5)$$

The corresponding hydraulic radius is

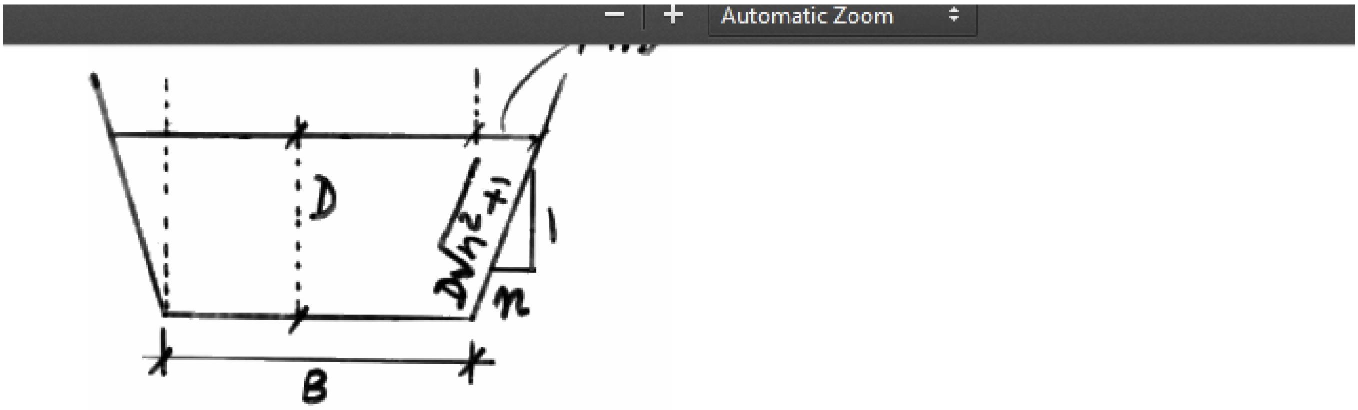
$$R_h = \frac{A}{P} = \frac{B \times D}{B + 2D} = \frac{2D \times D}{2D + 2D} = \frac{2D^2}{4D} \quad \text{and hence}$$

$$R_h = \frac{D}{2} \quad (7.6)$$

From equations (7.5) and (7.6), it is clear that rectangular channel will be most economical when either: (a) the depth of the flow is half the width (Eq. 7.5), or (b) the hydraulic radius is half the depth of flow (eq. 7.6).

1.2 Most Economical Trapezoidal Channel

Consider a trapezoidal section of channel as shown



Let B = width of channel at bottom, D = depth of flow, side slope = *1/n*

$\therefore$ area of flow,

$$A = \frac{\left[B + (B + 2nD)\right]D}{2} \quad \text{, or}$$

$$A = (B + nD)D \qquad (7.7a)$$

and wetted perimeter,

$$P = B + 2D\sqrt{1 + n^2} \qquad (7.7b)$$

from Eq. (7.7a) we have

$$B = \frac{A}{D} - nD \quad \text{which if substituted in (7.7b), we get}$$

$$P = \left(\frac{A}{D} - nD\right) + 2D\sqrt{1 + n^2} \qquad (7.7c)$$

For most economical cross section, P should be minimum or

$$\frac{dP}{dD} = 0, \qquad \text{so} \quad \frac{dP}{dD} = -\frac{A}{D^2} - n + 2\sqrt{1 + n^2} = 0 \quad \Rightarrow$$

$$2\sqrt{1 + n^2} = \frac{A}{D^2} + n \text{, using Eq. (7.7a) to replace } A \text{, we get}$$

$$2\sqrt{1 + n^2} = \frac{(B + nD)D}{D^2} + n = \frac{B + 2nD}{D}, \qquad \text{and hence}$$

$$D\sqrt{1 + n^2} = \frac{B + 2nD}{2} \qquad (7.8)$$

To obtain the corresponding hydraulic radius, we can use equation (7.8) to re-write equation (7.7b) as

$$P = B + B + 2nD = 2(B + nD) \qquad (7.9)$$

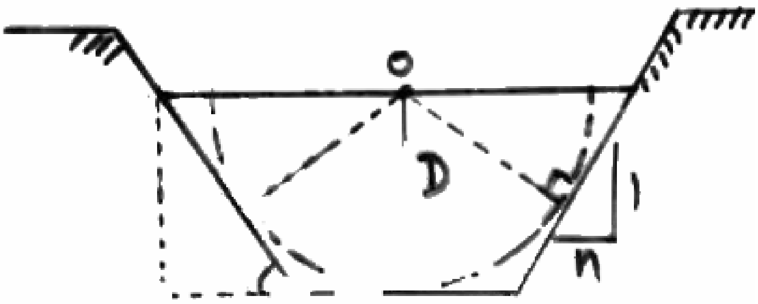
Equations (7.7a) and (7.9) are used to give the corresponding hydraulic radius:

$$R_h = \frac{A}{P} = \frac{(B + nD)D}{2(B + nD)} \quad \text{and hence}$$

$$R_h = \frac{D}{2} \qquad (7.10)$$

From equations (7.8) and (7.10), it is clear that trapezoidal channel will be most economical when either: (i) one of the sloping sides (wetted length) = half of the top width (Eq. 7.8), or (ii) the hydraulic radius is half the depth of flow (Eq. 7.10).

There is, however, a third condition that could be used to produce the most economical trapezoidal channel: “A trapezoidal section channel is most economical if when a semi-circle is drawn with its center, O, on the water surface and radius equal to the depth of flow, D, the three sides of the channel are tangential to the semi-circle”. To prove this condition, using the figure shown, we have:



$$OF = OM \sin\alpha = \frac{1}{2}(B + 2nD)\sin\alpha = (\frac{B}{2} + nD)\sin\alpha, \qquad \text{or}$$

$$\sin\alpha = \frac{OF}{\frac{B}{2} + nD} \qquad (7.11a)$$

and using triangle KMN, we have

$$\sin\alpha = \frac{MK}{MN} = \frac{D}{D\sqrt{1 + n^2}} \qquad (7.11b)$$

Comparing equations (7.11a) and (7.11b), we obtain

$$OF = \frac{(B/2) + nD}{\sqrt{1+n^2}}$$, using equation (7.8) to replace the numerator , we obtain

$$OF = \frac{D\sqrt{1+n^2}}{\sqrt{1+n^2}}$$, therefore;

$$OF = D \qquad (7.12)$$

Thus, if a semi-circle is drawn with O as center and radius equal to the depth of flow D, the three sides of a most economical trapezoidal section will be tangential to the semi-circle.

Best side slope for most economical trapezoidal section can be shown to be when

$$n = \frac{1}{\sqrt{3}}:$$

So far we assumed that the side slopes are constant. Let us now consider the case when the side slopes can also vary. The most economical side slopes of a most economical trapezoidal section can be obtained as follows:

Equation (7.8) can be re-written as

$$B = 2D(\sqrt{1+n^2} - n) \qquad (7.13a)$$

and from Eq. (7.7a) we have

$$B = \frac{A}{D} - nD \qquad (7.13b)$$

equating the above two equations, we get

$$\frac{A}{D} - nD = 2D(\sqrt{1+n^2} - n)$$ from which we find

$$D^2 = \frac{A}{2\sqrt{1+n^2} - n} \qquad (7.14)$$

Now, from equation (7.9), $P = 2(B + nD)$, and equation (7.13b), we can write

$$P = 2\frac{A}{D}$$, squaring both sides and use equation (7.14), we get

$$P^2 = 4(\frac{A}{D})^2 = 4A(2\sqrt{1+n^2} - n),$$

or most economical side slopes of a most efficient cross section we satisfy the condition $\frac{dP}{dn} = 0$, i.e.;

$$2P\frac{dP}{dn} = 4A\left[(1+n^2)^{-\frac{1}{2}} * (2n) - 1\right]$$ and applying the condition we get

$$\frac{2n}{\sqrt{1+n^2}} = 1$$, simplifying we get

$$4n^2 = 1 + n^2,$$

and hence

$$n = \frac{1}{\sqrt{3}} \quad \text{or}$$

$$\frac{1}{n} = \frac{\sqrt{3}}{1} = \tan\theta \quad \Rightarrow \quad \theta = 60^o$$

Therefore, best side slope is at 60_o to the horizontal, i.e.; of all trapezoidal sections a half hexagon is most economical. However, because of constructional difficulties, it may not be practical to adopt the most economical side slopes

1.3 Most Economical Circular Channel

As discussed in section 7.5 that for a most economical section the discharge, for a constant cross-sectional area, slope of bed and resistance coefficient, is maximum (or P is minimum). But in the case of circular channels, the area of the flow cannot be maintained constant. Indeed, the cross-sectional area A and the wetted perimeter P both do not depend on D but they depend on the angle α . Referring to the figure shown, we can determine the wetted perimeter P and the area of flow A as follows

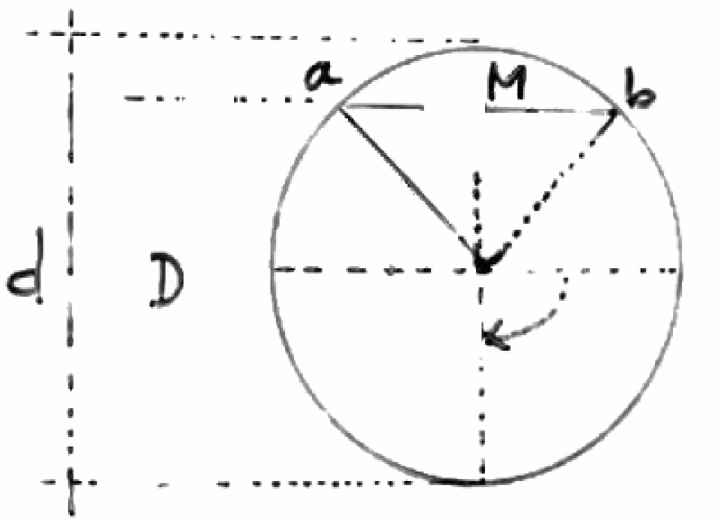
Let

D = depth of flow

d = diameter of pipe

r = radius of pipe

2α = angle subtended by the free surface at the center (in radians)



$$P = 2\alpha r = \alpha d \qquad (7.15)$$

$$A = A_1 + A_2$$

$$A_1 = \frac{1}{2}(2\alpha)\, r^2 = \alpha r^2 = \frac{\alpha d^2}{4}$$

$$A_2 = area\ of\ \Delta oab = \frac{1}{2} ab * OM = \frac{1}{2}(2 * Mb) * OM = Mb * OM$$

but $Mb = \frac{d}{2}\sin\theta = \frac{d}{2}\sin(180^o - \alpha) = \frac{d}{2}\sin\alpha$, and

$$OM = \frac{d}{2}\cos\theta = \frac{d}{2}\cos(180^o - \alpha) = -\frac{d}{2}\cos\alpha$$

therefore,

$$A_2 = -\frac{d^2}{4}\sin\alpha\cos\alpha = -\frac{d^2}{8}\sin 2\alpha,$$ and hence

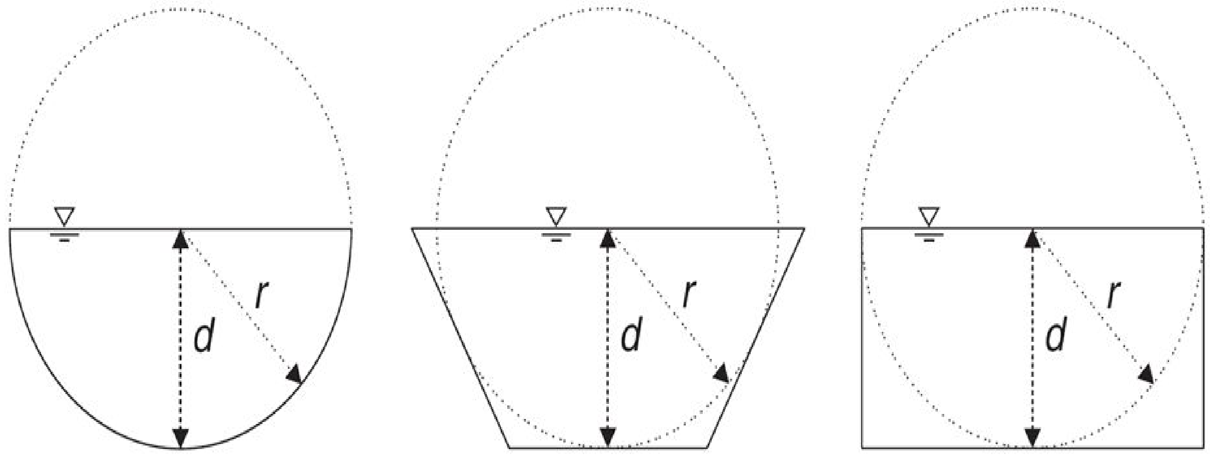
$$A = \frac{\alpha d^2}{4} - \frac{d^2}{8}\sin 2\alpha \qquad (7.16)$$

Consequently, the cross-sectional area A and the wetted perimeter P both depend on the angle α which is the most suitable variable. Thus in case of circular channels, for most economical section, two separate conditions are obtained: 1) condition for maximum discharge, and 2) condition for maximum velocity.

Most Economical Section of Channels

For several types of channel cross-sections the 'best' design is shown in Fig. 5.1 and summarized in the following table:

Cross-section	Optimum width B	Optimum cross-sectional area A	Optimum wetted perimeter P_w	Optimum hydraulic diameter D_H
Rectangular	$2d$	$2d^2$	$4d$	$2d$
Trapezoidal	$\frac{2}{\sqrt{3}}d$	$\sqrt{3}d^2$	$2\sqrt{3}d$	$2d$
Semi-circle	$2d$	$\frac{\pi}{2}d^2$	πd	$2d$



Circular channel Trapezoidal channel Rectangular channel

Self-Check -2	Written Test

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. What are the most economical flow channels?
2. Describe each type of economic flow channel?
   A. Horizontal
   B. Trapezoid
   C. Circular

Information sheet -2	Differentiate a partial flow chart is used to identify the depth of flow for maximum discharge and maximum velocity

5.2. Differentiate A partial flow chart used to identify the depth of flow for maximum discharge and maximum velocity

The variation of **flow velocity** within a cross-section complicates the hydraulic analysis, so the engineer ... Note that the **depth used** should be the actual **depth** of **flow**, ... exits over the **top** of the weir, and, as with orifices, the point of **maximum** contraction is ... : V-Notch Weir Coefficients of **Discharge —** .C = **Discharge** coefficient from... H = **Depth** off **low** above elevation of crest in feet (approach **velocity** shall be ... Weirs are generally **used** as measuring and hydraulic control devices. .... pipe 48 inches in diameter or smaller, and to a **maximum** of 6 inches for pipe. generally **flow** faster, whereas rivers are deeper, wider and more ... friction against the bottom and banks causes the **velocity** to decrease from a **maximum** ..... With the **velocity** obtained in terms of the water **depth** and **discharge** [u = Q/A ..... to hold and can be **used** to **determine** the changes in the **flow** across.

Flow Conveyance discharge Water travels downhill from points of higher energy to points of lower energy (unless forced to do otherwise) until it reaches a point of equilibrium, such as an ocean. This tendency is facilitated by the presence of natural conveyance channels such as brooks, streams, and rivers. The water's journey may also be aided by man-made structures such as drainage swales, pipes, culverts, and canals. Hydraulic concepts can be applied equally to both man-made structures and natural features. Area, Wetted Perimeter, and Hydraulic Radius The term area refers to the cross-sectional area of flow within a channel. When a channel has a consistent cross-sectional shape, slope, and roughness, it is called a prismatic channel. If the flow in a conveyance section is open to the atmosphere, such as in a culvert flowing partially full or in a river, it is said to be open-channel flow or free-surface flow. If a channel is flowing completely full, as with a water distribution pipe, it is said to be operating under full-flow conditions. Pressure flow is a special type of full flow in which forces on the fluid cause it to push against the top of the channel as well as the bottom and sides. These forces may result from, for example, the weight of a column of water in a backed-up sewer manhole or elevated storage tank. A section's wetted perimeter is defined as the portion of the channel in contact with the flowing fluid.

Maximum Velocity;- the velocity of a section is not constant throughout the cross-sectional area. Instead, it varies with location. The velocity is zero where the fluid is in contact with the conduit wall. So The variation of flow velocity within a cross-section complicates the hydraulic analysis, so the engineer usually simplifies the situation by looking at the average (mean) velocity of the section for analysis purposes. This average velocity is defined as the total flow rate divided by the cross-sectional area, and is in units of length per time. $V = Q/A$ where V = average velocity (m/s, ft/s) Q = flow rate (m^3/s, ft^3/s) A = area (m^2, ft^2) Steady Flow Speaking in terms of flow, the word steady indicates that a constant flow rate is assumed throughout an analysis. In other words, the flow velocity does not change with respect to time at a given location. For most hydraulic calculations, this assumption is reasonable. A minimal increase in model accuracy does not warrant the time and effort that would be required to perform an analysis with changing (unsteady) flows over time. When analyzing triver networks, storm sewers, and other collection systems in which it is desirable to vary the flow rate at different locations throughout the system, the network can often be broken into segments

Velocity partial flow discharge chart

There are some basic issues common to both the types are following

1. Shape of the cross section of the canal.

2. Side slope of the canal.

3. Longitudinal bed slope.

4. Permissible velocities - Maximum and Minimum.

5. Roughness coefficient.

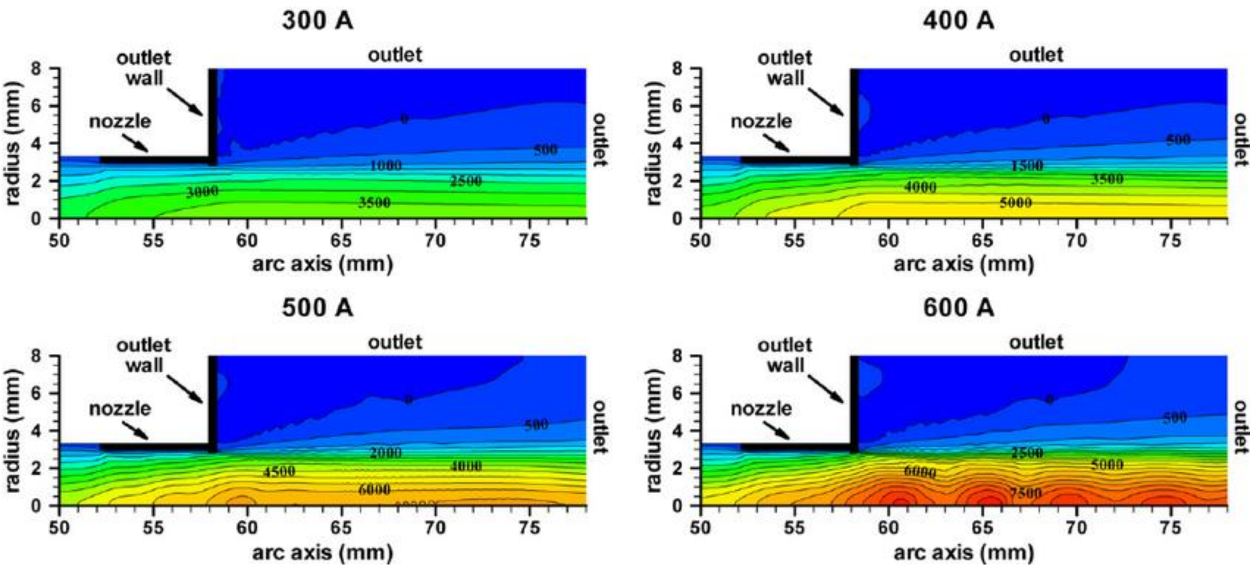
6. Free board.

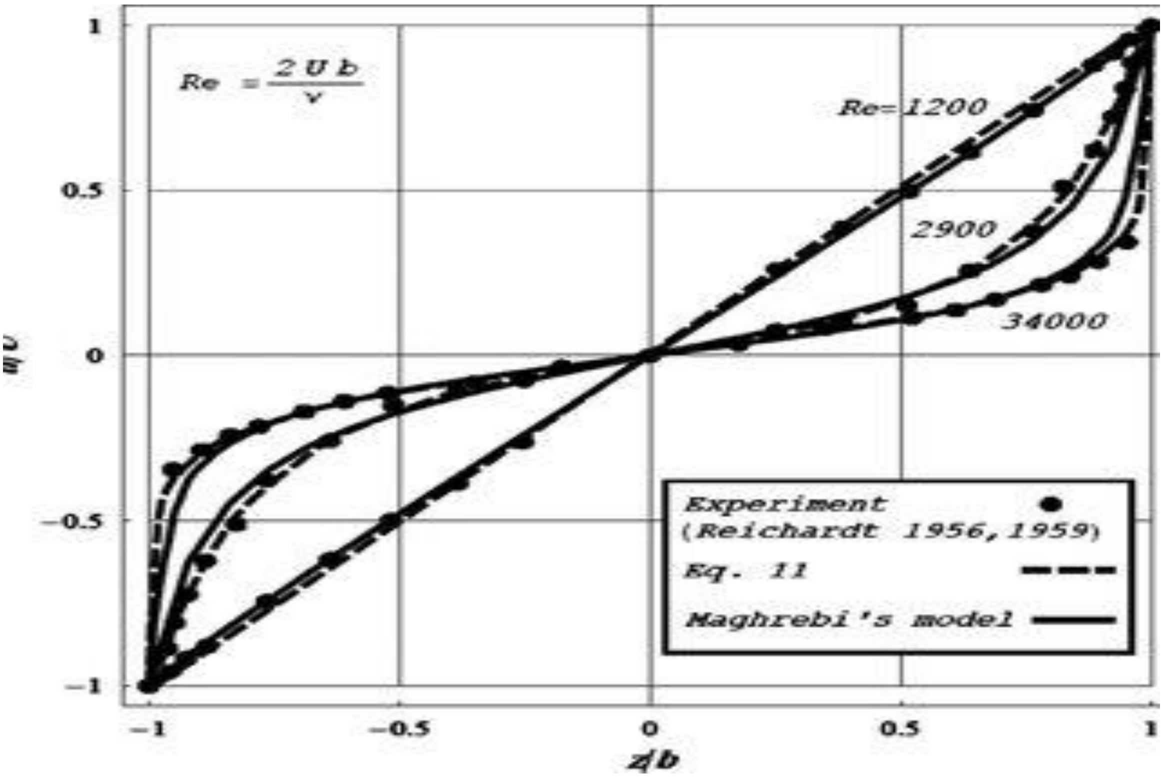
DISCHARGE CAPACITY IN m3/s Bank height for canals and free board for hard surface or buried membrane and, earth lining

Velocities: Minimum and Maximum It may be noted that canals carrying water with higher velocities may scour the bed and the sides of the channel leading to the collapse of the canal. On the other hand the weeds and plants grow in the channel when the nutrients are available in the water. Therefore, the minimum permissible velocity should not allow the growth of vegetation such as weed, hyacinth as well you should not be permitting the settlement of suspended material (non silting velocity). depend on the material that is used and the bed slope of the channel. For example: in case of chutes, spillways the velocity may reach as high as 25 m/s. As the dam heights are increasing the expected velocities of the flows are also increasing and it can reach as high as 70 m/s in exceptional cases. Thus, when one refers to maximum permissible velocity, it is for the normal canals built for irrigation purposes and Power canals in which the energy loss must be minimized.

Fig. identifying the depth of flow for maximum discharge and maximum velocity

Velocity contours in the outlet nozzle and near-discharge regions for 32.5 slm of argon. The partial characteristics radiation method is employed. Water mass flow rates are 0.228 g s − 1 (300 A), 0.315 g s − 1 (400 A), 0.329 g s − 1 (500 A), 0.363 g s − 1 (600 A), contour




Re = 2 U b / ν
Re=1200
2900
34000
Experiment (Reichardt 1956,1959)
Eq. 11
Maghrebi's model
u/U
z/b

Self-Check -2	Written Test

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1;-Explain most economical channel?

2;-What is the hydraulic diameter?

3;-what are flow control valves?

4;-what is difference b/n trapezoidal and rectangular chanels?