UNIT 3  LAMINAR AND TURBULENT FLOW

Orifice
Orifice is a small opening on the side or at the bottom of a tank, through which a fluid is flowing. The orifices are classified according to the size, shape, nature of discharge and shape of the edge.

1. According to the size of orifice and head of liquid from the centre of the orifice: Small orifice and Large orifice.
   Small Orifice: If the head of liquid from the centre of orifice is more than five times the depth of orifice, the orifice is called small orifice.
   Large Orifice: If the head of liquid is less than five times the depth of orifice, it is known as large orifice.

2. According to shape of orifice: (i) Circular orifice, (ii) Triangular orifice, (iii) Rectangular orifice and (iv) Square orifice

3. According to their cross-sectional area or edge: (i) Sharp-edged orifice and (ii) Bell mouthed orifice

4. According to the discharge condition: (i) Free discharging orifices (ii) Fully drowned or submerged orifices and (iii) Partially submerged orifices.

Flow through a Small Orifice
Flow from a tank through a hole in the side.

The edges of the hole are sharp to minimize frictional losses by minimizing the contact between the hole and the liquid. The streamlines at the orifice contract reducing the area of flow. This contraction is called the vena contracta.

The amount of contraction must be known to calculate the flow.

Applying Bernoulli’s equation along the streamline joining point 1 on the surface to point 2 at the centre of the orifice.
At the surface velocity is negligible \( v_1 = 0 \) and the pressure atmospheric \( p_1 = 0 \). At the orifice the jet is open to the atmosphere so again the pressure is atmospheric \( p_2 = 0 \).

If we take the datum line through the orifice then \( z_1 = H \) and \( z_2 = 0 \), leaving

\[
v_2 = v_{th} = \sqrt{2gH}
\]

This theoretical value of velocity is an overestimate as friction losses have not been taken into account. A coefficient of velocity is used to correct the theoretical velocity,

\[
v_a = C_v \times v_{th}
\]

Each orifice has its own coefficient of velocity, they usually lie in the range 0.97 - 0.99

The discharge through the orifice = jet area \( \times \) jet velocity
The area of the jet is the area of the vena contracta and not the area of the orifice.
We use a Coefficient of contraction to get the area of the jet, \( A_a \).
\[
A_a = C_v \times \text{area of orifice}
\]
Discharge through the orifice;
\[ Q = Av \]
Actual discharge, \( Q_a = C_d \times Q_{th} \)
\[ Q_{th} = \text{Area of orifice} \times v_{th} \]

**Hydraulic Coefficients**

The following three coefficients are known as *hydraulic coefficients* or *orifice coefficients*.

- Coefficient of contraction
- Coefficient of velocity
- Coefficient of discharge

**Coefficient Of Contraction**

The ratio of the area of the jet, at vena-contracta, to the area of the orifice is known as *coefficient of contraction*. Mathematically coefficient of contraction,

\[ C_c = \frac{\text{Area of jet at vena contracta}}{\text{Area of the orifice}} \]

The value of Coefficient of contraction varies slightly with the available head of the liquid, size and shape of the orifice. The average value of \( C_c \) is 0.64.

**Coefficient of Velocity**

The ratio of actual velocity of the jet, at vena-contracta, to the theoretical velocity is known as *coefficient of velocity*.

The theoretical velocity of jet at vena-contracta is given by the relation,

\[ v_{th} = \sqrt{2gH} \]

Mathematically coefficient of velocity,

\[ C_v = \frac{\text{Actual velocity of the jet at vena contracta}}{\text{Theoretical velocity of the jet}} \]

The difference between the velocities is due to friction of the orifice. The value of Coefficient of velocity varies slightly with the different shapes of the edges of the orifice. This value is very small for sharp-edged orifices. For a sharp edged orifice, the value of \( C_v \) increases with the head of water.

**Coefficient of Discharge**

The ratio of a actual discharge through an orifice to the theoretical discharge is known as *coefficient of discharge*. Mathematically coefficient of discharge,

\[ C_d = \frac{\text{Actual discharge}}{\text{Theoretical discharge}} \]
Thus the value of coefficient of discharge varies with the values of $C_C$ and $C_v$. An average of coefficient of discharge varies from 0.60 to 0.64.

**Determination of Coefficient of Discharge ($C_d$)**

The water is allowed to flow through an orifice provided in a tank under a constant head $H$. The water is collected in a collecting tank for a known height. The time of collection of water in the collecting tank is noted down.

Then

actual discharge through orifice, $Q_a = \frac{\text{Area of collecting tank}(A) \times \text{Height of water}(h)}{\text{time }(t)}$

and theoretical discharge, $Q_{th} = \text{area of orifice} \times \sqrt{2gH}$

$$C_d = \frac{Q_a}{Q_{th}}$$

**Determination of Coefficient of Velocity ($C_v$)**

In the jet issuing from an orifice under a constant head of $H$, let the coordinate distances be ‘$x$’ horizontally and ‘$y$’ vertically of a water particle be traced in ‘$t$’ seconds from vena contracta.

Actual velocity, $v_a = \sqrt{\frac{g x^2}{2y}}$ and theoretical velocity, $v_{th} = \sqrt{2gH}$

Coefficient of Velocity, $C_v = \frac{\sqrt{\frac{g x^2}{2y}}}{\sqrt{2gH}} = \sqrt{\frac{x^2}{4yH}}$

$$C_v = \sqrt{\frac{x^2}{4yH}}$$

**Discharge Through a Large Rectangular Orifice**

Consider a large rectangular orifice of width ‘$b$’ provided on a vertical side of a tank, with a constant head ‘$H$’ above its centre, discharging freely in to atmosphere.
Let, \( H_1 \) = head of water above bottom edge of the orifice and \( H_2 \) = head of water above the top edge of orifice. Therefore, the height of orifice is \( (H_1 - H_2) \).

\( C_d \) = co-efficient of discharge.

Consider a thin horizontal strip of depth \( 'dh' \) and \( 'h' \) be the head of water above the strip.

Area of the strip = \( b \, dh \); theoretical velocity of water through the strip = \( \sqrt{2gh} \)

Discharge through elementary strip = \( b \, dh \, \sqrt{2gh} \)

Total discharge through the entire orifice = \( C_d \int_{H_2}^{H_1} b \, dh \, \sqrt{2gh} \)

\[
Q_a = C_d b \sqrt{2g} \int_{H_2}^{H_1} dh \, h^{\frac{3}{2}}
= C_d b \sqrt{2g} \left[ \frac{h^{3/2}}{\frac{2}{3}} \right]_{H_2}^{H_1}
\]

\[
Q_a = \frac{2}{3} C_d b \sqrt{2g} \left[ H_1^{\frac{3}{2}} - H_2^{\frac{3}{2}} \right] \text{ m}^3/\text{s}
\]

**Discharge through Fully Submerged Orifice**

If the discharge side of a large rectangular orifice is under water, then it is known to be fully submerged or drowned Orifice.

Let, \( 'b' \) is the width of orifice;
\( H_1 = \) head of water above bottom edge of the orifice and \( H_2 = \) head of water above the top edge of orifice.

Therefore, the height of orifice is \( (H_1 - H_2) \)

Head difference between the upstream and downstream side water levels which is responsible for the flow = \( H \) metres.

\( C_d \) = co-efficient of discharge.

Discharge, \( Q_a = C_d \times \text{area} \times \text{velocity} = C_d \, b(H_1 - H_2)\sqrt{2gH} \)

**Discharge through Partially Submerged Orifice**

If part of the discharge end of the rectangular orifice is submerged under water, it is known as partially submerged orifice and the discharge is through two parts.

Upper part is a free orifice and lower part is submerged orifice.
Let, ‘b’ is the width of orifice;
H₁ = head of water above bottom edge of the orifice and H₂ = head of water above the top edge of orifice.
Head difference between the upstream and downstream side water levels which is responsible for the flow = H metres.

Total discharge partially submerged orifice, Q = Q₁ + Q₂
Q₁= Discharge through the submerged orifice part
Q₂= Discharge through the free orifice part

\[ Q₁ = C_d \cdot b(H₁ - H)\sqrt{2gH} \]
\[ Q₂ = \frac{2}{3} C_d b \sqrt{2g} \left[ \frac{H^3}{H_{2}} - \frac{H^3}{H_{2}} \right] \]

\[ Q = C_d \cdot b(H₁ - H)\sqrt{2gH} + \frac{2}{3} C_d b \sqrt{2g} \left[ \frac{H^3}{H_{2}} - \frac{H^3}{H_{2}} \right] \quad m^3/s \]

1. The head of water over an orifice of diameter 50 mm is 12 m. Find the actual discharge and actual velocity of jet at vena contracta. Take Cd = 0.6 and Cv = 0.98.

Solution: Diameter of orifice, d = 50 mm = 0.05 m
Head, H = 12 m
Theoretical velocity, \( v_{th} = \sqrt{2gH} = \sqrt{2 \times 9.81 \times 12} = 15.34 \text{ m/s} \)
Actual velocity, \( v_a = C_v \times v_{th} = 0.98 \times 15.34 = 15.03 \text{ m/s} \)
Actual discharge, \( Q_a = C_d \times Q_{th} = C_d \times \sqrt{2gH} = 0.6 \times \frac{\pi}{4} \times d^2 \times 15.34 = 0.6 \times \frac{\pi}{4} \times 0.05^2 \times 15.34 \]
\[ = 0.01807 \text{ m}^3/\text{s} \]

2. The head of water over the centre of an orifice of diameter 30 mm is 1.5 m. The actual discharge through the orifice is 2.35 litres/sec. Find the co-efficient of discharge.

Actual discharge, \( Q_a = 2.35 \text{ lit/s} = 0.00235 \text{ m}^3/\text{s} \)
Theoretical discharge, \( Q_{th} = a \sqrt{2gH} = \frac{\pi}{4} \times d^2 \times \sqrt{2gH} \)
\[ Q_{th} = \frac{\pi}{4} \times 0.03^2 \times \sqrt{2 \times 9.81 \times 1.5} = 0.003835 \text{ m}^3/\text{s} \]
Co-efficient of discharge, \( C_d = \frac{Q_a}{Q_{th}} = 0.00235/0.003835 = 0.613 \)

3. A jet of water, issuing from a sharp edged vertical orifice under a constant head of 60 cm, has the horizontal and vertical co-ordinates measured from the vena contracta at a certain point as 10 cm and 0.45 cm respectively. Find the value of Cv. Also find the value of Cc if Cd = 0.6

Solution:
\[ C_v = \sqrt{\frac{x^2}{4yH}} = \sqrt{\frac{0.1^2}{4 \times 0.0045 \times 0.6}} = 0.96 \]
We know that, \( C_d = C_c \times C_v \)
\( C_c = C_d / C_v = 0.6 / 0.96 = 0.625 \)
4. The head of water over an orifice of diameter 100 mm is 5 m. The water coming out from orifice is collected in a circular tank of diameter 2 m. The rise of water level in circular tank is 0.45 m in 30 seconds. Also the coordinates of a certain point on the jet, measured from vena contracta are 100 cm horizontal and 5.2 cm vertical. Find the hydraulic co-efficients.

Solution:
Actual discharge, \( Q_a = \text{Area of collecting tank} \times \text{height of collection/time} \)
\[ Q_a = \frac{\pi D^2}{4} \times h/t = \frac{\pi}{4} \times 2^2 \times 0.45/30 = 0.047124 \text{ m}^3/\text{s} \]

Theoretical discharge, \( Q_{th} = a \sqrt{2gH} = \frac{\pi}{4} \times d^2 \times \sqrt{2gH} \)
\[ Q_{th} = \frac{\pi}{4} \times 0.1^2 \times \sqrt{2} \times 9.81 \times 5 = 0.0778 \text{ m}^3/\text{s} \]

Co-efficient of discharge, \( C_d = \frac{Q_a}{Q_{th}} = 0.047124/0.0778 = 0.61 \)

\( C_v = \sqrt{\frac{x^2}{4yH}} = \sqrt{\frac{1^2}{4 \times 0.052 \times 5}} = 0.98 \)

\( C_c = C_d/C_v = 0.61/0.98 = 0.62 \)

5. A rectangular orifice, 2 m wide and 1.5 m deep is discharging water from a tank. If the water level in the tank is 3 m above the top edge of the orifice, find the discharge through the orifice. Take \( C_d = 0.6 \).

\( H_1 = \text{head of water above bottom edge of the orifice} = 1.5 + 3 = 4.5 \text{ m} \)
\( H_2 = \text{head of water above the top edge of orifice} = 3 \text{ m} \)

Discharge through the free orifice, \( Q_a = \frac{2}{3} C_d b \sqrt{2gH} \left[ H_1^3 - H_2^3 \right] \)
\[ = \frac{2}{3} \times 0.6 \times 2 \times \sqrt{2 \times 9.81} \times \left[ 4.5^3 - 3^3 \right] \text{ m}^3/\text{s} \]
\[ = 15.414 \text{ m}^3/\text{s} \]

6. Find the discharge through a fully submerged orifice of width 2 m if the difference of water levels on both the sides of the orifice be 800 mm. The height of water from top and bottom of the orifice are 2.5 m and 3 m respectively. Take \( C_d = 0.62 \).

Discharge, \( Q_a = C_d b (H_1 - H_2) \sqrt{2gH} = 0.62 \times 2 \times (3 - 2.5) \times \sqrt{2} \times 9.81 \times 0.8 = 2.46 \text{ m}^3/\text{s} \)

7. A rectangular orifice of 1.5 m wide and 1.2 m deep is provided in one side of a large lank. The water level on one side of the orifice is 2 m above the top edge of the orifice, while on the other side of the orifice the water level is 0.4 m below its top edge. Calculate the discharge through the orifice if \( C_d = 0.62 \)

Solution:
\( H_1 = \text{head of water above bottom edge of the orifice} = 1.2 + 2 = 3.2 \text{ m} \)
\( H_2 = \text{head of water above the top edge of orifice} = 2 \text{ m} \)
\( H = \text{Head difference} = 2 + 0.4 = 2.4 \text{ m} \)

Total discharge partially submerged orifice, \( Q = Q_1 + Q_2 \)
\( Q_1 = \text{Discharge through the submerged orifice part} \)
Q₂= Discharge through the free orifice part

\[ Q = C_d b (H_1 - H) \sqrt{2gH} + \frac{2}{3} C_d b \sqrt{2g} \left[ H_{\frac{3}{2}} - H_{\frac{3}{2}}^2 \right] m^3/s \]

\[ = 0.62 \times 1.5 \times (3.2 - 2.4) \times \sqrt{2 \times 9.81 \times 2.4} + \frac{2}{3} \times 0.62 \times 1.5 \times \sqrt{2 \times 9.81} \times \left[ 2.4^{\frac{3}{2}} - 2^3 \right] \]

\[ = 5.1054 + 2.4432 = 7.5486 \text{ m}^3/\text{s} \]

**Flow Over Notches and Weirs**

A *notch* is a device used for measuring the rate of flow of a liquid through a small channel. A *weir* is a concrete or masonry structure placed in the open channel over which the flow occurs.

**Nappe and crest**

The sheet of water flowing through a notch or over a weir is known as nappe or vein. The bottom edge of the notch or the top of a weir over which water flows is known as sill or crest. The height above the bottom of the tank or channel is known as crest height.

**Fig. Nappe and crest**

**Difference between orifice and notch**

<table>
<thead>
<tr>
<th>Orifice</th>
<th>Notch</th>
</tr>
</thead>
<tbody>
<tr>
<td>An orifice may be defined as an opening provided in the side or bottom of tank or vessel such that the liquid flows through the entire orifice.</td>
<td>A notch may be defined as an opening provided in the side of tank or vessel such that the liquid surface in tank is below the top edge of opening.</td>
</tr>
</tbody>
</table>

**Difference between notches and weirs**

<table>
<thead>
<tr>
<th>Notch</th>
<th>Weir</th>
</tr>
</thead>
</table>
A notch may be defined as an opening provided in the side of tank or vessel such that the liquid surface in tank is below the top edge of opening. A weir may be defined as any regular obstruction in open stream over which the flow takes place.

<table>
<thead>
<tr>
<th>Small structure</th>
<th>Large structure</th>
</tr>
</thead>
<tbody>
<tr>
<td>Made of metallic plates.</td>
<td>Made of concrete/bricks.</td>
</tr>
<tr>
<td>Measure small flow rate.</td>
<td>Measure large flow rate.</td>
</tr>
</tbody>
</table>

Types of Notches

<table>
<thead>
<tr>
<th>Types of notches</th>
<th>Diagram</th>
<th>Discharge/flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>a. Rectangular</td>
<td><img src="image" alt="Rectangular Notch Diagram" /></td>
<td>[ Q = \frac{2}{3} C_d L \sqrt{2g} \left( H \right)^{3/2} ]</td>
</tr>
<tr>
<td>b. Triangular</td>
<td><img src="image" alt="Triangular Notch Diagram" /></td>
<td>[ Q = \frac{8}{15} C_d \sqrt{2g} \tan \frac{\theta}{2} \left( H \right)^{5/2} ]</td>
</tr>
<tr>
<td>c. Trapezoidal</td>
<td><img src="image" alt="Trapezoidal Notch Diagram" /></td>
<td>[ Q = \frac{2}{3} C_{d1} L \sqrt{2g} \left( H \right)^{3/2} + \frac{8}{15} C_{d2} \sqrt{2g} \tan \frac{\theta}{2} \left( H \right)^{5/2} ]</td>
</tr>
</tbody>
</table>
| d. Stepped        | ![Stepped Notch Diagram](image) | \[ Q_1 = \frac{2}{3} C_d L_1 \sqrt{2g} \left( H_1 \right)^{3/2} \]
|                  |         | \[ Q_2 = \frac{2}{3} C_d L_2 \sqrt{2g} \left[ \left( H_2 \right)^{3/2} - \left( H_1 \right)^{3/2} \right] \]
|                  |         | \[ Q_3 = \frac{2}{3} C_d L_2 \sqrt{2g} \left[ \left( H_3 \right)^{3/2} - \left( H_2 \right)^{3/2} \right] \] |

Types of Weir
1. **Shape**
   - Rectangular
   - Triangular
   - Trapezoidal

**Types of weir on basis of shape**

<table>
<thead>
<tr>
<th>Types of weir</th>
<th>Diagram</th>
<th>Discharge/flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>a. Rectangular</td>
<td><img src="image1" alt="Diagram" /></td>
<td>( Q = \frac{2}{3} C_d L \sqrt{2g} (H)^{3/2} )</td>
</tr>
<tr>
<td>b. Triangular</td>
<td><img src="image2" alt="Diagram" /></td>
<td>( Q = \frac{8}{15} C_d \sqrt{2g} \tan \frac{\theta}{2} (H)^{5/2} )</td>
</tr>
<tr>
<td>c. Trapezoidal</td>
<td><img src="image3" alt="Diagram" /></td>
<td>( Q = Q_1 + Q_2 ) ( Q = \frac{2}{3} C_{d1} L \sqrt{2g} (H)^{3/2} + \frac{8}{15} C_{d2} \sqrt{2g} \tan \frac{\theta}{2} (H)^{5/2} )</td>
</tr>
</tbody>
</table>

**Note:** The discharge equation for rectangular, triangular and trapezoidal weir is same as of notch.

2. **Nature of discharge**
   - Free: Liquid level on the downstream side is lower than the crest.
     ![Free flowing weir](image4)
     
   - Drowned: Liquid level submerges the crest
     ![Drowned weir](image5)

3. **Width of crest**
   - Sharp: The crest is narrow
     ![Sharp crest weir](image6)
Broad: The crest is broad

1. Determine flow rate through a rectangular notch if \( L = 0.20 \) m and \( H = 0.15 \) m. Take value of \( c_d = 0.645 \)

Solution

\[
Q = \frac{2}{3} c_d L \sqrt{2g} (H)^{3/2}
\]

\[
= \frac{2}{3} \times 0.645 \times 0.2 \times \sqrt{2} \times 9.81 \times (0.15)^{3/2}
\]

\[
= 0.022 \text{ m}^3/\text{s}
\]

2. Determine flow rate through a triangular notch if \( H = 0.20 \) m and \( \theta = 60^\circ \). Take value of \( c_d = 0.85 \).

Solution

\[
Q = \frac{8}{15} c_d \sqrt{2g} \tan \frac{\theta}{2} (H)^{5/2}
\]

\[
= \frac{8}{15} \times 0.85 \times \sqrt{2} \times g \times \tan \frac{60}{2} (0.20)^{5/2}
\]

\[
= 0.0207 \text{ m}^3/\text{s}
\]

3. Determine flow rate through a rectangular weir if \( L = 0.30 \) m and \( H = 0.25 \) m. Take value of \( c_d = 0.95 \)

Solution

\[
Q = \frac{2}{3} c_d L \sqrt{2g} (H)^{3/2}
\]

\[
= \frac{2}{3} \times 0.95 \times 0.3 \times \sqrt{2} \times 9.81 \times (0.25)^{3/2}
\]

\[
= 0.105 \text{ m}^3/\text{s}
\]
Flow in pipes
In this chapter, however, a method of expressing the loss using an average flow velocity is stated. Studies will be made on how to express losses caused by a change in the cross sectional area of a pipe, a pipe bend and a valve, in addition to the frictional loss of a pipe. Consider a case where fluid runs from a tank into a pipe whose entrance section is fully rounded. At the entrance, the velocity distribution is roughly uniform while the pressure head is lower by V²/2g. As shown in below Figure, the section from the entrance to just where the boundary layer develops to the tube centre is called the inlet or entrance region, whose length is called the inlet or entrance length.

For steady flow at a known flow rate, these regions exhibit the following:

**Laminar flow**: A local velocity constant with time, but which varies spatially due to viscous shear and geometry.

**Turbulent flow**: A local velocity which has a constant mean value but also has a statistically random fluctuating component due to turbulence in the flow. Typical plots of velocity time histories for laminar flow, turbulent flow, and the region of transition between the two are shown below.

Principal parameter used to specify the type of flow regime is the Reynolds number:

\[ \text{Re} = \frac{\rho V D}{\mu} = \frac{V D}{\nu} \]

V - characteristic flow velocity
D - characteristic flow dimension
\( \mu \) - dynamic viscosity
\( \nu \) - kinematic viscosity

We can now define the critical or transition Reynolds number \( \text{Re}_{cr} \). \( \text{Re} \) is the Reynolds number below which the flow is laminar, above which the flow is turbulent. While transition can occur over a range of \( \text{Re} \), we will use the following for internal pipe or duct flow:

**Major Losses due to pipe Friction**
Let us study the flow in the region where the velocity distribution is fully developed after passing through the inlet region as shown below. If a fluid is flowing in the round pipe of diameter \( d \) at the average flow velocity \( v \), let the pressures at two points distance \( L \) apart be \( p_1 \) and \( p_2 \) respectively. The relationship between the velocity \( u \) and the loss head \( h = (p_1 - p_2)/\rho g \) For the laminar flow, the loss head \( h \) is proportional to the flow velocity \( v \) while for the turbulent
flow, it turns out to be proportional to \( v^{1.75-2} \).
The loss head is expressed by the following equation as shown in this equation:

\[
h_f = f \frac{L}{d} \frac{V^2}{2g}
\]

This equation is called the Darcy-Weisbach equation', and the coefficient \( f \) is called the friction coefficient of the pipe.

**Minor losses in pipes**
In a pipe line, in addition to frictional loss, head loss is produced through additional turbulence arising when fluid flows through such components as change of area change of direction, branching, junction, bend and valve.

1. **Head Loss due to sudden expansion**
For a suddenly expanding pipe as shown in below Figure, assume that the pipe is horizontal, disregard the frictional loss of the pipe, let \( h \) be the expansion loss, and set up an equation of energy between sections 1 and 2 as:

\[
\frac{p_1}{\rho g} + \frac{v_1^2}{2g} = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + h
\]

\[
h_1 = \frac{p_1 - p_2}{\rho g} + \frac{v_1^2 - v_2^2}{2g}
\]

\[
h_e = \frac{(v_1 - v_2)^2}{2g}
\]

2. **Head loss at the exit of pipes**:
\[
h_{ex} = \frac{v^2}{2g}
\]

\( v \) = velocity of liquid in the pipe

3. **Head loss due to sudden contraction**
Owing to the inertia, section 1 (section area \( A_1 \)) of the fluid shrinks to section 2 (section area \( A_c \)) and then widens to section 3 (section area \( A_2 \)). The loss when the flow is accelerated is extremely small, followed by ahead loss similar to that in the case of sudden expansion equation

![Diagram of fluid flow through contraction](image-url)
\[ h_c = \frac{(v_c - v_t)^2}{2g} = \left( \frac{A_2}{A_c} - 1 \right)^2 \frac{v_t^2}{2g} = \left( \frac{1}{C_c} - 1 \right)^2 \frac{v_t^2}{2g} \]

\[ h_c = 0.5 \frac{v_t^2}{2g} \]

Here \( C_c = \frac{A_c}{A_2} \) is a contraction coefficient.

4. **Head loss at Inlet of pipe**
   The loss of head in the case where fluid enters into a pipe from a large vessel is expressed by the following equation:

\[ h_i = 0.5 \frac{v^2}{2g} \]

5. **Head loss due to bend in pipe**
   \[ h_b = k \frac{v^2}{2g} \]
   where \( k \) = Coefficient of pipe bend; \( v \) = mean velocity of liquid in the pipe.

6. **Head loss due to various pipe fittings**
   \[ h_p = k \frac{v^2}{2g} \]
   where \( k \) = Coefficient of pipe fittings

7. **Head loss due to obstruction**
   \[ h_o = \left( \frac{A}{C_c(A - a)} \right)^2 \frac{v^2}{2g} \]

   where, \( A \) = Area of pipe cross section
   \( a \) = area of obstruction; \( v \) = velocity of liquid

**Pipes in series**
when pipes of different diameters are connected end to end to form a pipe line, they are said to be in series. The total loss of energy (or head) will be the sum of the losses in each pipe plus minor losses at connections.

Example
Consider the two reservoirs shown in figure, connected by a single pipe that changes diameter over its length. The surfaces of the two reservoirs have a difference in level of 9m. The pipe has a diameter of 200mm for the first 15m (from A to C) then a diameter of 250 mm for the remaining 45m (from C to B)

The join at C is sudden. For both pipes use
Total head loss for the system $H = \text{height difference of reservoirs}$

$h_{f1} = \text{head loss for 200mm diameter section of pipe}$

$h_{f2} = \text{head loss for 250mm diameter section of pipe}$

$h_i = \text{head loss at entry point}$

$h_j = \text{head loss at join of the two pipes}$

$h_{ex} = \text{head loss at exit point}$

So

$H = hf_1 + hf_2 + hi + hj + hex = 9 \text{m}$

All losses are, in terms of $Q$

$$h_{f1} = \frac{fLQ^2}{2d_i^2}$$

$$h_{f2} = \frac{fLQ^2}{2d_2^2}$$

$$h_{i_{1,2}} = \frac{0.5}{2g} \left( \frac{Q}{2\pi d^2} \right)^2 = 0.5 \times 0.0826 \frac{Q^2}{d^4} = 0.0413 \frac{Q^2}{d^4}$$

$$h_{ex} = 1.0 \times \frac{Q^2}{d^2} = 0.0826 \frac{Q^2}{d^2}$$

$$h_{i_{2,3}} = \left( \frac{u_i - u_j}{2g} \right) \left( \frac{1}{d_i} - \frac{1}{d_2} \right)^2 = 0.0826Q^2 \left( \frac{1}{d_i^2} - \frac{1}{d_2^2} \right)^2$$

Substitute these into

$hf_1 + hf_2 + hi + hj + hex = 9 \text{m}$

and solve for $Q$, to give $Q = 0.158 \text{ m}^3/\text{s}$

**Pipes in parallel**

When two or more pipes in parallel connect two reservoirs, as shown in Figure 17, for example, then the fluid may flow down any of the available pipes at possible different rates. But the head difference over each pipe will always be the same. The total volume flow rate will be the sum of the flow in each pipe.

The analysis can be carried out by simply treating each pipe individually and summing flow rates at the end.

**Pipes in Parallel Example**

Two pipes connect two reservoirs (A and B) which have a height difference of 10m. Pipe 1 has diameter 50mm and length 100m. Pipe 2 has diameter 100mm and length 100m. Both have entry loss $k_L = 0.5$ and exit loss $k_L = 1.0$ and Darcy $f$ of 0.008.

Calculate:
a) rate of flow for each pipe
b) the diameter D of a pipe 100m long that could replace the two pipes and provide the same flow.

using the Darcy equation in terms of Q

\[ h_f = \frac{fLQ^2}{3d^4} \]

And the loss equations in terms of Q:

\[ h_l = k \frac{u^2}{2g} = k \frac{Q^2}{2gA^2} = k \frac{A^2}{2g\pi^2} \frac{Q^2}{d^4} = 0.0826k \frac{Q^2}{d^4} \]

For pipe 1

\[ 10 = h_{entry} + hf + h_{exit} \]
\[ 10 = 0.0826 \times 0.5 \frac{Q^2}{0.05^4} + 0.008 \times 100 \frac{Q^2}{3 \times 0.05^5} + 0.0826 \times 1.0 \frac{Q^2}{0.05^4} \]
\[ Q = 0.0034 \text{ m}^3 / \text{s} \]
\[ Q = 3.4 \text{ litres} / \text{s} \]

For pipe 2

\[ 10 = h_{entry} + hf + h_{exit} \]
\[ 10 = 0.0826 \times 0.5 \frac{Q^2}{0.1^4} + 0.008 \times 100 \frac{Q^2}{3 \times 0.1^5} + 0.0826 \times 1.0 \frac{Q^2}{0.1^4} \]
\[ Q = 0.0188 \text{ m}^3 / \text{s} \]
\[ Q = 18.8 \text{ litres} / \text{s} \]

**BOUNDARY LAYER FLOWS**

Viscous internal flows have the following major boundary layer characteristics:

* An entrance region where the boundary layer grows and dP/dx ≠ constant,

* A fully developed region where:
  * The boundary layer fills the entire flow area.
  * The velocity profiles, pressure gradient, and \( \frac{\partial w}{\partial x} \) are constant; i.e. they are not equal to \( f(x) \),
  * The flow is either laminar or turbulent over the entire length of the flow, i.e. transition from laminar to turbulent is not considered.

However, viscous flow boundary layer characteristics for external flows are significantly different as shown below for flow over a flat plate:
Schematic of boundary layer flow over a flat plate

For these conditions, we note the following characteristics:

- The boundary layer thickness grows continuously from the start of the fluid-surface contact, e.g. the leading edge. It is a function of x, not a constant.

- Velocity profiles and shear stress are \( f(x,y) \).

- The flow will generally be laminar starting from \( x = 0 \).

- The flow will undergo laminar-to-turbulent transition if the streamwise dimension is greater than a distance \( x_{cr} \) corresponding to the location of the transition Reynolds number \( Re_{cr} \).

- Outside of the boundary layer region, free stream conditions exist where velocity gradients and therefore viscous effects are typically negligible.

As it was for internal flows, the most important fluid flow parameter is the local Reynolds number defined as

\[
Re_x = \frac{\rho U_x x}{\mu} = \frac{U_x x}{\nu}
\]

where

- \( \rho \) = fluid density
- \( \mu \) = fluid dynamic viscosity
- \( \nu \) = fluid kinematic viscosity
- \( U_x \) = characteristic flow velocity
- \( x \) = characteristic flow dimension

It should be noted at this point that all external flow applications will not use a distance from the leading edge \( x \) as the characteristic flow dimension. For example, for flow over a cylinder, the diameter will be used as the characteristic dimension for the Reynolds number.

Transition from laminar to turbulent flow typically occurs at the local transition Reynolds number, which for flat plate flows can be in the range of

\[
500,000 \leq Re_{cr} \leq 3,000,00
\]
With $x_{cr}$ = the value of $x$ where transition from laminar to turbulent flow occurs, the typical value used for steady, incompressible flow over a flat plate is

$$Re_{cr} = \frac{\rho U_{\infty} x_{cr}}{\mu} = 500,000$$

Thus for flat plate flows for which:

$x < x_{cr}$ the flow is laminar

$x \geq x_{cr}$ the flow is turbulent

The solution to boundary layer flows is obtained from the reduced “Navier – Stokes” equations, i.e., Navier-Stokes equations for which boundary layer assumptions and approximations have been applied.

**Flat Plate Boundary Layer Theory**

**Laminar Flow Analysis**

For steady, incompressible flow over a flat plate, the laminar boundary layer equations are:

Conservation of mass:  \[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \]

'X' momentum:  \[ u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{d p}{d x} + \frac{1}{\rho} \frac{\partial}{\partial y} \left( \mu \frac{\partial u}{\partial y} \right) \]

'Y' momentum:  \[ -\frac{\partial p}{\partial y} = 0 \]

The solution to these equations was obtained in 1908 by Blasius, a student of Prandtl's. He showed that the solution to the velocity profile, shown in the table below, could be obtained as a function of a single, non-dimensional variable defined as

Table: Blasius Velocity Profile
\[ \eta = y \left( \frac{U_\infty}{U_X} \right)^{1/2} \]

with the resulting ordinary differential equation:

\[ f''' + \frac{1}{2} f f'' = 0 \]

and \[ f'(\eta) = \frac{u}{U_\infty} \]

Fluid flow

- Fluid flow over bodies frequently occurs in practice and physical phenomena:
  - Drag force acting on automobile, power lines, trees and underwater pipeline
  - The lift development by airplane wings
  - Upward draft of rain, snow and dust particles in high winds
  - The transportation of red blood cells by blood flow
  - The vibration and noise generated by bodies moving in a fluid
  - Power generated by wing turbines and...etc

Drag

- The force flowing fluid exerts on a body in the flow direction is called drag.
- The drag force can be measured directly by simply attaching the body subjected to fluid flow to a calibrated spring and measuring the displacement in flow direction (like measuring weight with a spring scale). Example: using drag-measuring device called drag balance-using flexible beams fitted with strain gages to measure the drag electronically.
- Drag usually an desireble effect, like friction \( \rightarrow \) we need to minimize it
- Reduction of drag associated with:
  - Reduction of fuel consumption in automobile, aircraft and submarines
  - Improved safety and durability of structures subjected to high winds and
  - Reduction of noise and vibration
  - Both of pressure force and wall shear force on the surface in the direction of flow is called **drag force**.
- The components of the pressure and wall shear force in the direction normal to the flow tend to move body in that direction and its sum is called **lift**.
- shear stress and pressure integrated over body surface
- **drag**: force component in the direction of upstream velocity
- **lift**: force normal to upstream velocity
Boundary Layers
When a fluid flows over a stationary surface, e.g. the bed of a river, or the wall of a pipe, the fluid touching the surface is brought to rest by the shear stress $\tau$ at the wall. The velocity increases from the wall to a maximum in the main stream of the flow.

Looking at this two-dimensionally we get the above velocity profile from the wall to the centre of the flow.

This profile doesn't just exit, it must build up gradually from the point where the fluid starts to flow past the surface - e.g. when it enters a pipe.

If we consider a flat plate in the middle of a fluid, we will look at the build up of the velocity profile as the fluid moves over the plate.

Upstream the velocity profile is uniform, (free stream flow) a long way downstream we have the velocity profile we have talked about above. This is the known as **fully developed flow**. But how do we get to that state?

This region, where there is a velocity profile in the flow due to the shear stress at the wall, we call the **boundary layer**. The stages of the formation of the boundary layer are shown in the figure below:
Formation of the boundary layer

Above we noted that the boundary layer grows from zero when a fluid starts to flow over a solid surface. As it passes over a greater length more fluid is slowed by friction between the fluid layers close to the boundary. Hence the thickness of the slower layer increases.

The fluid near the top of the boundary layer is dragging the fluid nearer to the solid surface along. The mechanism for this dragging may be one of two types:

The first type occurs when the normal viscous forces (the forces which hold the fluid together) are large enough to exert drag effects on the slower moving fluid close to the solid boundary. If the boundary layer is thin then the velocity gradient normal to the surface, \( \frac{du}{dy} \), is large so by Newton's law of viscosity the shear stress, \( t = \mu \left( \frac{du}{dy} \right) \), is also large. The corresponding force may then be large enough to exert drag on the fluid close to the surface.

As the boundary layer thickness becomes greater, so the velocity gradient become smaller and the shear stress decreases until it is no longer enough to drag the slow fluid near the surface along. If this viscous force was the only action then the fluid would come to a rest.

It, of course, does not come to rest but the second mechanism comes into play. Up to this point the flow has been laminar and Newton's law of viscosity has applied. This part of the boundary layer is known as the laminar boundary layer.

The viscous shear stresses have held the fluid particles in a constant motion within layers. They become small as the boundary layer increases in thickness and the velocity gradient gets smaller. Eventually they are no longer able to hold the flow in layers and the fluid starts to rotate.

This causes the fluid motion to rapidly becomes turbulent. Fluid from the fast moving region moves to the slower zone transferring momentum and thus maintaining the fluid by the wall in motion. Conversely, slow moving fluid moves to the faster moving region slowing it down. The net effect is an increase in momentum in the boundary layer. We call the part of the boundary layer the turbulent boundary layer.

At points very close to the boundary the velocity gradients become very large and the velocity gradients become very large with the viscous shear forces again becoming large enough to maintain the fluid in laminar motion. This region is known as the laminar sub-layer. This layer occurs within the turbulent zone and is next to the wall and very thin - a few hundredths of a mm.

Surface roughness effect

Despite its thinness, the laminar sub-layer can play a vital role in the friction characteristics of the surface.
This is particularly relevant when defining pipe friction - as will be seen in more detail in the level 2 module. In **turbulent** flow if the height of the roughness of a pipe is greater than the thickness of the laminar sub-layer then this increases the amount of turbulence and energy losses in the flow. If the height of roughness is less than the thickness of the laminar sub-layer the pipe is said to be **smooth** and it has little effect on the boundary layer.

**In laminar** flow the height of roughness has very little effect

**Boundary layers in pipes**

As flow enters a pipe the boundary layer will initially be of the laminar form. This will change depending on the ration of inertial and viscous forces; i.e. whether we have laminar (viscous forces high) or turbulent flow (inertial forces high).

From earlier we saw how we could calculate whether a particular flow in a pipe is laminar or turbulent using the Reynolds number.

\[
Re = \frac{\rho ud}{\mu}
\]

\( (r = \text{density} \ u = \text{velocity} \ m = \text{viscosity} \ d = \text{pipe diameter}) \)

Laminar flow: \( Re < 2000 \)

Transitional flow: \( 2000 < Re < 4000 \)

Turbulent flow: \( Re > 4000 \)

If we only have laminar flow the profile is parabolic - as proved in earlier lectures - as only the first part of the boundary layer growth diagram is used. So we get the top diagram in the above figure.
If turbulent (or transitional), both the laminar and the turbulent (transitional) zones of the boundary layer growth diagram are used. The growth of the velocity profile is thus like the bottom diagram in the above figure.

Once the boundary layer has reached the centre of the pipe the flow is said to be fully developed. (Note that at this point the whole of the fluid is now affected by the boundary friction.)

The length of pipe before fully developed flow is achieved is different for the two types of flow. The length is known as the entry length.

Laminar flow entry length 120 diameter

Turbulent flow entry length 60 diameter

**Boundary layer separation**

Convergent flows: Negative pressure gradients

If flow over a boundary occurs when there is a pressure decrease in the direction of flow, the fluid will accelerate and the boundary layer will become thinner.

This is the case for convergent flows.

The accelerating fluid maintains the fluid close to the wall in motion. Hence the flow remains stable and turbulence reduces. Boundary layer separation does not occur.

Divergent flows: Positive pressure gradients

When the pressure increases in the direction of flow the situation is very different. Fluid outside the boundary layer has enough momentum to overcome this pressure which is trying to push it backwards. The fluid within the boundary layer has so little momentum that it will very quickly be brought to rest, and possibly reversed in direction. If this reversal occurs it lifts the boundary layer away from the surface as shown below.
This phenomenon is known as **boundary layer separation**.

At the edge of the separated boundary layer, where the velocities change direction, a line of vortices occur (known as a vortex sheet). This happens because fluid to either side is moving in the opposite direction.

This boundary layer separation and increase in the turbulence because of the vortices results in very large energy losses in the flow.

These separating / divergent flows are inherently unstable and far more energy is lost than in parallel or convergent flow.

**Examples of boundary layer separation**

A divergent duct or diffuser

The increasing area of flow causes a velocity drop (according to continuity) and hence a pressure rise (according to the Bernoulli equation).
Increasing the angle of the diffuser increases the probability of boundary layer separation. In a Venturi meter it has been found that an angle of about 6 provides the optimum balance between length of meter and danger of boundary layer separation which would cause unacceptable pressure energy losses.

**Tee-Junctions**

Assuming equal sized pipes, as fluid is removed, the velocities at 2 and 3 are smaller than at 1, the entrance to the tee. Thus the pressure at 2 and 3 are higher than at 1. These two adverse pressure gradients can cause the two separations shown in the diagram above.

**Y-Junctions**

Tee junctions are special cases of the Y-junction with similar separation zones occurring. See the diagram below.

Downstream, away from the junction, the boundary layer reattaches and normal flow occurs i.e. the effect of the boundary layer separation is only local. Nevertheless fluid downstream of the junction will have lost energy.

**Bends**
Two separation zones occur in bends as shown above. The pressure at b must be greater than at a as it must provide the required radial acceleration for the fluid to get round the bend. There is thus an adverse pressure gradient between a and b so separation may occur here.

Pressure at c is less than at the entrance to the bend but pressure at d has returned to near the entrance value - again this adverse pressure gradient may cause boundary layer separation.

Flow past a cylinder

The pattern of flow around a cylinder varies with the velocity of flow. If flow is very slow with the Reynolds number \( (r \times v \times \text{diameter/m}) \) less than 0.5, then there is no separation of the boundary layers as the pressure difference around the cylinder is very small. The pattern is something like that in the figure below.

If \( 2 < \text{Re} < 70 \) then the boundary layers separate symmetrically on either side of the cylinder. The ends of these separated zones remain attached to the cylinder, as shown below.

Above a Re of 70 the ends of the separated zones curl up into vortices and detach alternately from each side forming a trail of vortices on the down stream side of the cylinder. This trial in known as a **Karman vortex trail** or **street**. This vortex trail can easily be seen in a river by looking over a bridge where there is a pier to see the line of vortices flowing away from the bridge. The phenomenon is responsible for the whistling of hanging telephone or power cables. A more significant event was the famous failure of the Tacoma narrows bridge. Here the frequency of the alternate vortex shedding matched the natural frequency of the bridge deck and resonance amplified the vibrations until the bridge collapsed. (The frequency of vortex shedding from a cylinder can be predicted. We will not try to predict it here but a derivation of the expression can be found in many fluid mechanics text books.)
Looking at the figure above, the formation of the separation occurs as the fluid accelerates from the centre to get round the cylinder (it must accelerate as it has further to go than the surrounding fluid). It reaches a maximum at Y, where it also has also dropped in pressure. The adverse pressure gradient between here and the downstream side of the cylinder will cause the boundary layer separation if the flow is fast enough, (Re > 2.)

Aerofoil

Normal flow over a aerofoil (a wing cross-section) is shown in the figure below with the boundary layers greatly exaggerated.

The velocity increases as air it flows over the wing. The pressure distribution is similar to that shown below so transverse lift force occurs.

If the angle of the wing becomes too great and boundary layer separation occurs on the top of the aerofoil the pressure pattern will change dramatically. This phenomenon is known as **stalling**.
When stalling occurs, all, or most, of the 'suction' pressure is lost, and the plane will suddenly drop from the sky! The only solution to this is to put the plane into a dive to regain the boundary layer. A transverse lift force is then exerted on the wing which gives the pilot some control and allows the plane to be pulled out of the dive.

Fortunately there are some mechanisms for preventing stalling. They all rely on preventing the boundary layer from separating in the first place.

1. Arranging the engine intakes so that they draw slow air from the boundary layer at the rear of the wing though small holes helps to keep the boundary layer close to the wing. Greater pressure gradients can be maintained before separation take place.
2. Slower moving air on the upper surface can be increased in speed by bringing air from the high pressure area on the bottom of the wing through slots. Pressure will decrease on the top so the adverse pressure gradient which would cause the boundary layer separation reduces.
3. Putting a flap on the end of the wing and tilting it before separation occurs increases the velocity over the top of the wing, again reducing the pressure and chance of separation occurring.

**Moody diagram**

**Turbulent Flow and the Moody Diagram:**
*Turbulent flow* is a flow regime in which the movement of the fluid particles is chaotic, eddying, and unsteady. According to table 3.1 (in your book), turbulent flow occurs at Re > 4000. Due to the complex nature of turbulent flows, scientists and engineers use empirical rather than theoretical approaches to model and design processes and machinery involving fluids.

As an initial approach to ‘empirical approximations’ to fluid flow in conduits, we shall utilize the dimensional analysis approach described in previous lectures. It was earlier found that for an incompressible fluid flow in a straight, horizontal, circular pipe of constant cross-sectional area, the significant variables are: that the wall shear stress $\tau_w$, the distance from the inlet $(x)$, pipe diameter $(D)$, flow average velocity $(u_m)$, fluid density $(\rho)$, fluid viscosity $(\mu)$, and the wall roughness $(\varepsilon)$. Dimensional analysis resulted with the following relationship:
\[
\frac{\tau_w}{\rho u_m^2} = \psi \left( \text{Re}, \frac{\epsilon}{D} \right)
\]

The function \( \psi \), which varies with the relative roughness \( \frac{\epsilon}{D} \) and Reynolds number is designated \( f \), the friction factor.

Expressing the above relationship in terms of \( f \), we have:

\[
f = \frac{\tau_w}{\rho u_m^2}
\]

Other versions in more common use are the *Fanning* friction factor:

\[
f_F = \frac{\tau_w}{\frac{1}{2} \rho u_m^2}
\]

**Piping and Pumping Problems**

Upward flow in an inclined pipe is depicted in Fig. 3.9 (below). A steady-state momentum balance in the direction of flow on the fluid in the pipe gives:

![Pressure drop in an inclined pipe.](image)

\[
(P_1 - P_2) \frac{\pi D^2}{4} - \tau_w \pi D L - \frac{\pi D^2}{4} \rho L g \sin \theta = 0
\]

Rearranging the substituting for wall shear stress in terms of the friction factor \( f \) gives:

\[
-\Delta P = 2 f_F \rho u_m^2 \frac{L}{D} + \rho g \Delta z = \frac{32 f_F \rho Q^2 L}{\pi^2 D^3} + \rho g \Delta z
\]

An alternative and somewhat more generally useful form of the above equation is obtained by investigating the frictional dissipation per unit mass. Rearrangement of the preceding equation yields:

\[
g \Delta z + \frac{\Delta P}{\rho} + 2 f_F u_m^2 \frac{L}{D} = 0
\]

But the overall (incompressible) energy balance for fluid flow in a pipe is:
\[ g \Delta z + \frac{\Delta P}{\rho} + \Delta \left( \frac{u^2}{2} \right) + F + w = 0 \]

Realizing that the change in kinetic energy is zero (constant cross section) and there is no work involved, then by comparing the preceding two equations, we obtain:

\[ F = 2f_F u_m^2 \frac{L}{D} = \frac{32f_F \rho Q^2 L}{\pi^2 D^5} \]

A result that is valid for LAMINAR or TURBULENT flow.

For the special case of LAMINAR flow, from the analysis obtained earlier (Hagen-Poiseuille equation), we know that:

\[ F = 2f_F u_m^2 \frac{L}{D} = \frac{8 \mu u_m L}{\rho a^2} \]

Since \( a = \frac{D}{2} \), the friction factor for laminar flow is therefore:

\[ f_F = \frac{16 \mu}{\rho u_m D} = \frac{16}{Re} \]

Experimentally, the friction factor depends on the Re and (if turbulent) on the pipe relative roughness. The relationship between the friction factor, Re, and relative roughness is schematically presented in Figure. This diagram is generally referred to as the *Moody Diagram*.

**Example**
Water (T= 20 °C) flows at a rate of 0.05 m$^3$/s in a 20 cm asphalted cast-iron pipe. What is the head loss (frictional losses) per kilometer of pipe?

Solution

1. Calculate the average flow velocity using the continuity equation
2. Calculate the Reynolds number (to determine whether the flow is laminar or turbulent)
3. Calculate the relative roughness
4. Look up the $f$ (friction factor) using the Moody Diagram

Average velocity:

$$u_m = \frac{Q}{A} = \frac{0.05 \text{ m}^3/\text{s}}{\frac{\pi}{4} (0.02 \text{ m})^2} = 1.59 \text{ m/s}$$

Reynolds number:

$$Re = \frac{u_m D}{v} = \frac{\left(1.59 \text{ m/s}\right) (0.20 \text{ m})}{10^{-6} \text{ m}^2/\text{s}} = 3.1 \times 10^5$$

Friction factor ($f$):

Pipe material equivalent roughness (asphalted cast-iron pipe) = 0.12 mm

Relative roughness =

$$\frac{\varepsilon}{D} = \frac{0.0012 \text{ m}}{0.2 \text{ m}} = 0.006$$

$f_r$ from diagram ~ 0.00475

Frictional loss:

$$F = 2f_r u_m^2 \frac{L}{D} = 2(0.00475) \left(1.59 \text{ m/s}\right)^2 \frac{1000 \text{ m}}{0.20 \text{ m}} = 120 \text{ m}^2/\text{s}^2 = 120 \frac{\text{J}}{\text{kg}}$$