BERNOULLI EQUATION

The motion of a fluid is usually extremely complex.

The study of a fluid at rest, or in relative equilibrium, was simplified by the absence of shear stress, but when a fluid flows over a solid surface or other boundary, whether stationary or moving, the velocity of the fluid in contact with the boundary must be the same that the boundary, and a velocity gradient is created at right angle to the boundary.

The resulting change of velocity from layer to layer of fluid flowing parallel to the boundary gives rise to shear stresses in the fluid.

Individual particles of fluid move as a result of the action of forces set up by differences of pressure of elevation.

Their motion is controlled by their inertia and the effect of the shear stresses exerted by the surrounding fluid.

The resulting motion is not easily analysed mathematically, and it is often necessary to supplement theory by experiment.
Motion of Fluid Particles and Streams

1. **Streamline** is an imaginary curve in the fluid across which, at a given instant, there is no flow.

![Streamline](image)

Figure 1

2. **Steady flow** is one in which the velocity, pressure and cross-section of the stream may vary from point to point but do not change with time.

   If, at a given point, conditions do change with time, the flow is described as **unsteady flow**.

3. **Uniform flow** occur if the velocity at a given instant is the same in magnitude and direction at every point in the fluid.

   If, at the given instant, the velocity changes from point to point, the flow is described as **non-uniform flow**.
Figure 2

At time T1

(i) Steady Flow and uniform

At time T2

(ii) Steady Flow and non-uniform

(iii) Unsteady Flow and uniform

(iv) Unsteady flow and non-uniform
4. **Real fluid** is a fluid which when it flows past a boundary, the fluid immediately in contact with the boundary will have the same velocity as the boundary.

**Ideal fluid** is a fluid which is assumed to have no viscosity and in which there are no shear stresses.

![Figure 3](image-url)
5. **Compressible fluid** is a fluid which its density will change with pressure.

6. **Laminar flow**, sometimes known as streamline flow, occurs when a fluid flows in parallel layers, with no disruption between the layers.

**Turbulent flow** is a flow regime characterized by chaotic, stochastic property changes.

From the observation done by Osborne Reynolds in 1883, in straight pipes of constant diameter, flow can be assumed to be turbulent if the Reynolds number, Re, exceeds 4000.

\[
\text{Re} = \frac{\rho v D}{\mu}
\]
Bernoulli Equation

![Diagram of Bernoulli's principle](image)

Mass per unit time flowing;
\[ = \rho Av \]

Rate of increase of momentum from AB to CD;
\[ = \rho Av[(v + \delta v) - v] \]
\[ = \rho Av \delta v \]

Force due to pressure at surface AB;
\[ = pA \]
Force due to pressure at surface CD;
\[ = (p + \delta p)(A + \delta A) \]

Force due to pressure at side surface;
\[ = p_{\text{side}} \delta A \] (can be neglected)

Force due to weight of the component;
\[ = mg \cos \theta \\
= \rho g V \\
= \rho g (A + \frac{1}{2} A) \delta s \cdot \frac{\delta z}{\delta s} \]

Neglecting products of small quantities.

Resultant force in the direction of motion
\[ = -A \delta p - \rho g A \delta z \]

Applying the Newton’s second law;
\[ \rho A v \delta v = -A \delta p - \rho g A \delta z \]

Dividing by \( \rho A \delta s \)
\[ 0 = \frac{1}{\rho} \frac{\delta p}{\delta s} + v \frac{\delta v}{\delta s} + g \frac{\delta z}{\delta s} \]
In the limit as $\delta s \to 0$
\[
0 = \frac{1}{\rho} \frac{dp}{ds} + \nu \frac{dv}{ds} + g \frac{dz}{ds}
\] (eq.1)

This is known as Euler’s equation, giving, in differential form, the relationship between pressure, velocity, density and elevation along a streamline for steady flow.

It cannot be integrated until the relationship between density and pressure is known.

**For an incompressible fluid**, for which density is constant, integration of Euler’s equation (eq.1) along the streamline, with respect to $s$, gives;

\[
\text{constant} = \frac{p}{\rho} + \frac{v^2}{2} + gz
\]

It can be written as;

\[
\text{constant} = \frac{p}{\rho g} + \frac{v^2}{2g} + z
\] (eq.2)

It is called Bernoulli equation.
To use it correctly, we must constantly remember the basic assumptions used in its derivation:

1. Viscous effect are assumed negligible
2. The flow is assumed to be steady
3. The flow is assumed to be incompressible
4. The equation is applicable along a streamline

If Bernoulli equation (eq.2) is integrated along the streamline between any two points indicated by suffixes 1 and 2;

\[
\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2
\]  

(eq.3)
Physical Interpretation

Integration of the equation of motion to give eq.2 actually corresponds to the work-energy principle often used in the study of dynamics. The work done on a particle by all forces acting on the particle is equal to the change of the kinetic energy of the particle.

Each of the terms in this equation has the units of energy per weight (LF/F = L) or length (feet, meters) and represents a certain type of head.

The elevation term, z, is related to the potential energy of the particle and is called the elevation head.

The pressure term, p/ρg, is called the pressure head and represents the height of a column of the fluid that is needed to produce the pressure p.

The velocity term, V^2/2g, is the velocity head and represents the vertical distance needed for the fluid to fall freely (neglecting friction) if it is to reach velocity V from rest.

The Bernoulli equation states that the sum of the pressure head, the velocity head, and the elevation head is constant along a streamline.
Static, Stagnation, Dynamic and Total Pressure

The second term in the Bernoulli equation, $V^2/2g$, is termed the dynamic pressure.

Its interpretation can be seen in Figure 6 by considering the pressure at the end of a small tube inserted into the flow and pointing upstream.

After the initial transient motion has died out, the liquid will fill the tube to a height of $H$ as shown. The fluid in the tube, including that at its tip, (2), will be stationary. That is, $V_2 = 0$, or point (2) is a stagnation point.
If we apply the Bernoulli equation between points (1) and (2), using $V_2 = 0$ and assuming that $z_1 = z_2$, we find that

$$p_2 = p_1 + \frac{1}{2} \rho v_1^2$$

Hence, the pressure at the stagnation point is greater than the static pressure, $p_1$, by an amount $\frac{1}{2} \rho v_1^2$, the dynamic pressure.
It can be shown that there is a stagnation point on any stationary body that is placed into a flowing fluid.

Some of the fluid flows “over” and some “under” the object. The dividing line (or surface for two-dimensional flows) is termed the stagnation streamline and terminates at the stagnation point on the body.

For symmetrical objects (such as a sphere) the stagnation point is clearly at the tip or front of the object as shown in Figure 7(a).

For nonsymmetrical objects such as the airplane shown in Figure 7(b), the location of the stagnation point is not always obvious.
Knowledge of the values of the static and stagnation pressures in a fluid implies that the fluid speed can be calculated.

This is the principle on which the Pitot-static tube is based H. de Pitot (1695–1771), as shown in Figure 8.
Consider a fluid flowing through a fixed volume that has one inlet and one outlet as shown in Figure 1.

If the flow is steady so that there is no additional accumulation of fluid within the volume, the rate at which the fluid flows into the volume must equal the rate at which it flows out of the volume.

Otherwise, mass would not be conserved. The mass flowrate from an outlet is given as below:

\[ \dot{m} = \rho Q = \rho AV \]

\( \dot{m} \): Mass flowrate  
\( Q \): Volume flowrate  
\( A \): Outlet area  
\( V \): Average velocity
To conserve mass, the inflow rate must equal the outflow rate. If the inlet is designated as (1) and the outlet as (2), it follows that; 
\[ \dot{m}_1 = \dot{m}_2 \]

Thus, conservation of mass requires; 
\[ \rho_1 A_1 V_1 = \rho_2 A_2 V_2 \]

If the density remains constant, then \( \rho_1 = \rho_2 \), 
And the above equation becomes the continuity equation for incompressible flow, and shown as; 
\[ A_1 V_1 = A_2 V_2 \]

or 
\[ Q_1 = Q_2 \]
EXAMPLE OF USE OF THE BERNOULLI EQUATION

Free jets

![Diagram of fluid motion](image)

Figure 1

From the fact, we found that;

\[ z_1 = h \quad \text{and} \quad z_2 = 0 \]
\[ p_1 = p_2 = 0 \quad \text{and} \quad v_1 = 0 \]

Thus, the fluid leaves as a “free jets” with;

\[ v_2 = \sqrt{2gh} \]

This is introduced in 1643 by Torricelli (1608-1647)
Nozzle

We can safely use the centerline velocity at point (2) as a reasonable “average velocity”, as shown in Figure 2(a).

If the exit is not a smooth, well-contoured nozzle, but rather a flat plate as shown in Figure 2(b), the diameter of the jet, \(d_j\) will be less that the diameter of the hole, \(d_h\).

This phenomenon is called a vena contracta effect, is a result of the inability of the fluid to turn the sharp 90-degree corner indicated by the dotted line in the figure.
The vena contracta effect is a function of the geometry of the outlet. It can be obtained by experimental, and called as contraction coefficient, $C_c$.

$$C_c = \frac{A_j}{A_h} = (\frac{d_j}{d_h})^2$$

with $A_j$ is area of the jet
$A_h$ is area of the hole
The pitot-static tube

Figure 4

The specific gravity of the manometer fluid shown in Figure 4 is 1.07. Determine the volume flowrate, $Q$, if the flow is inviscid and incompressible and the flowing fluid is water.
The orifice nozzle / The nozzle meter

Determine the flowrate through the submerged orifice shown in Figure 5 if the contraction coefficient is $C_c = 0.63$
The venturi meter

JP-4 fuel ($SG=0.77$) flows through the Venturi meter shown in Figure 6. Determine the elevation, $h$, of the fuel in the open tube connected to the throat of the Venturi meter.
A rectangular weir

The volume flowrate, \( Q \), follows that;

\[
Q = C_1 H b \sqrt{2gH} = C_1 b (H)^{\frac{3}{2}} (\sqrt{2g})
\]
A triangular weir

![Diagram of a triangular weir]

The volume flowrate, \( Q \), follows that;

\[
Q = C_2 \left( \frac{1}{2} \tan \theta \right) (H)^{\frac{5}{2}} \left( \sqrt{2g} \right)
\]
The energy line and the hydraulic grade line

As discussed before, the Bernoulli equation is actually an energy equation representing the partitioning of energy for an inviscid, incompressible, steady flow.

The sum of the various energies of the fluid remains constant as the fluid flows from one section to another.

A useful interpretation of the Bernoulli equation can be obtained through the use of the concepts of the hydraulic grade line (HGL) and the energy line (EL).

This ideas represent a geometrical interpretation of a flow and can often be effectively used to better grasp the fundamental processes involved.
The energy line is a line that represents the total head available to the fluid. The elevation of the energy line can be obtained by measuring the stagnation pressure with a pitot tube.

The static pressure tap connected to the piezometer tube measures the sum of the pressure head and elevation head, and called piezometer head.

The locus provided by a series of piezometer taps is termed the hydraulic line.
Chapter 5 – Fluid in Motion – Examples of use of the Bernoulli equation.

Figure 10
KINEMATICS OF FLUID MOTION

The Velocity Field

The representation of properties of fluid parameters as function of the spatial coordinates is termed a field representation of the flow.

One of the most important fluid variables is the velocity field.

\[ V = u(x, y, z, t) \hat{i} + v(x, y, z, t) \hat{j} + w(x, y, z, t) \hat{k} \]

Where \( u, v \) and \( w \) are the \( x, y \) and \( z \) components of the velocity vector.

The speed of fluid is:

\[ V = |V| = \sqrt{u^2 + v^2 + w^2} \]
Example 1

The velocity field of a flow is given by
\[ V = (5z - 3)i + (x + 4)j + (4y)k \] (m/s)
Determine the fluid speed at the origin \((x=y=z=0)\)
And on the \(x\)-axis \((y=z=0)\)

Example 2

The velocity field of a flow is given by
\[ V = \left( \frac{20y}{\sqrt{x^2 + y^2}} \right)i - \left( \frac{20x}{\sqrt{x^2 + y^2}} \right)j \] (m/s)
Determine the fluid speed at points along the \(x\)-axis and along the \(y\)-axis.
What is the angle between the velocity vector and the \(x\)-axis at point \((x,y)=(5,0), (5,5)\) and \((0,5)\)

Example 3

The components of a velocity vector field are given by \(u=x+y, v=xy^3+16\) and \(w=0\).
Determine the location of any stagnation points \((V=0)\) in the flow field.
Eulerian and Lagrangian Flow Descriptions

There are two general approaches in analyzing fluid mechanics problems.

The first method is called the Eulerian method, the second method is called the Lagragian method.

From Eulerian method we obtain information about the flow in terms of what happens at fixed points in space as the fluid flows past those points.

Lagragian method involves following individual fluid particles as they move about and determining how the fluid properties associated with these particles change as a function of time. That is, the fluid particles are “tagged” or identified, and their properties determined as they move.
Steady and Unsteady Flow

We have assumed steady flow is the velocity at a given point in space does not vary with time.

\[
\frac{\partial V}{\partial t} = 0
\]

In reality, almost all flows are unsteady in some sense.

The unsteady flows are usually more difficult to analyze and to investigate experimentally than are steady flows.

Laminar flow – smooth flow

Turbulent flow – irregular flow
Streamlines

Streamline is a line that is everywhere tangent to the velocity field.

Streamlines are obtained analytically by integrating the equations defining lines tangent to the velocity field as illustrated in the Figure 3.

\[
\frac{dy}{dx} = \frac{v}{u}
\]
Streaklines and pathlines

A streakline consists of all particles in a flow that have previously passed through a common point.

Streaklines are more of a laboratory tool than an analytical tool. They can be obtained by taking instantaneous photographs of marked particles that all passed through a given location in the flow field at some earlier time.

A pathline is the line traced out by a given particle as it flows from one point to another. The pathline is a Lagrangian concept that can be produced in the laboratory by marking a fluid particle and taking a time exposure photograph of its motion.

Figure 4
The Acceleration Field

For the infrequently used Lagrangian method, we described the fluid acceleration just as is done in solid body dynamics, \( a = a(t) \) for each particle.

For the Eulerian description we describe the acceleration field as a function of position and time without actually following any particular particle.

This is analogous to describing the flow in terms of the velocity field,

\[
V = V(x, y, z, t)
\]

The acceleration of a particle is the time rate of change of its velocity.
The Material Derivative

Consider a fluid particle moving along its pathline as shown in Figure 5.

In general, the particle’s velocity, denoted \( V_A \) for particle A, is a function of its location and the time. That is:

\[
V_A = V_A(r_A, t) = V_A[x_A(t), y_A(t), z_A(t), t]
\]
Then, the acceleration field from the velocity field for any particle obtained as:

$$a = \frac{\partial V}{\partial t} + u \frac{\partial V}{\partial x} + v \frac{\partial V}{\partial y} + w \frac{\partial V}{\partial z}$$

This is a result whose scalar components can be written as:

\((x\text{-axis})\)

$$a_x = \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z}$$

\((y\text{-axis})\)

$$a_y = \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z}$$

\((z\text{-axis})\)

$$a_z = \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z}$$
Convective Effects

The portion of the material derivative represented by the spatial derivatives is termed the convective derivative.

It represents the fact that a flow property associated with a fluid particle may vary because of the motion of the particle from one point in space where the parameter has one value to another point in space where its value is different.

That portion of the acceleration is termed the convective acceleration.

For example, the temperature of a water particle changes as it flows through a water heater.

Figure 6
Example 4
The x and y components of a velocity field are given by \( u = x^2 y \) and \( v = -xy^2 \). Determine the equation for the streamlines of this flow.

Example 5
A velocity field is given by \( u = cx^2 \) and \( v = cy^2 \), where \( c \) is a constant. Determine the x and y components of the acceleration. At what point (points) in the flow is the acceleration zero.

Example 6
Determine the acceleration field for a three-dimensional flow with velocity components, \( u = -x \), \( v = 4x^2 y^2 \) and \( w = x - y \).
**Example 7**

A hydraulic jump is a rather sudden change in depth of a liquid layer as it flows in an open channel as shown in Figure 7. If $V_1=0.4\text{m/s}$, $V_2=0.1\text{m/s}$ and $\ell=0.07\text{m}$, estimate the average deceleration of the liquid as it flows across the hydraulic jump. How many G’s deceleration does this represent?

**Example 8**

$V_0=40\text{m/s}$ and $V=\frac{3}{2}V_0\sin\theta$. Determine the streamwise and normal components of acceleration at point $A$ if the radius of the sphere is $a=0.20\text{m}$. 
Streamline Coordinates

In many flow situations, it is convenient to use a coordinate system defined in terms of the streamlines of the flow.

The flows can be described either in $(x, y)$ Cartesian coordinate or $(r, \theta)$ polar coordinate system.
Control Volume and System Representations

A control volume, on the other hand, is a volume in space (a geometric entity, independent of mass) through which fluid may flow.

A system is a specific, identifiable quantity of matter. It may consist of a relatively large amount of mass, or it may be an infinitesimal size.

The system may interact with its surroundings by various means. It may continually change size and shape, but it always contains the same mass.
KINEMATICS OF FLUID MOTION

Reynolds Transport Theorem

We are sometimes interested in what happens to a particular part of the fluid as it moves about.

Other times we may be interested in what effect the fluid has on a particular object or volume in space as fluid interact with it.

Thus, we need to describe the laws governing fluid motion using both system concept (consider a given mass of the fluid) and control volume concept (consider a given volume).

To do this we need an analytical tool to shift from one representation to the other.

The Reynolds Transport Theorem provides this tool.
All physical laws are stated in terms of various physical parameters.

Velocity, acceleration, mass, temperature and momentum are but a few of the more common parameters.

Let $B$ represent any of these (or other) fluid parameters and $b$ represent the amount of that parameter per unit mass. That is:

$$B = m \cdot b$$

where $m$ is the mass of the portion of fluid of interest.

The $B$ is termed an **extensive property** and $b$ is termed an **intensive property**.

If $B$ is mass, it follows that $b$ is equal to 1.

If $B$ is the kinetic energy of mass, $B = m \frac{V^2}{2}$, then $b = \frac{V^2}{2}$, the kinetic energy per mass.
General form of the Reynolds transport theorem for a fixed, non-deforming control volume is given as ;

\[
\frac{DB_{sys}}{Dt} = \frac{\partial}{\partial t} \int_{cv} \rho b dV + \int_{cs} \rho b V \cdot \hat{n} \cdot dA
\]
Derivation of the Continuity Equation

A system is defined as a collection of unchanging contents, so the conservation of mass principle for a system is simply stated as;

\[
\text{Time rate of change of the system mass} = 0
\]

Or

\[
\frac{DM_{\text{sys}}}{Dt} = 0
\]

where the system mass, \( M_{\text{sys}} \), is more generally expressed as;

\[
M_{\text{sys}} = \int \rho \cdot dV
\]

and the integration is over the volume of the system.

In words, we can say that the system mass is equal to the sum of all the density-volume element products for the contents of the system.
The Reynolds transport theorem states that:

\[
\frac{D}{Dt} \int_{\text{sys}} \rho \cdot dV = \frac{\partial}{\partial t} \int_{\text{cv}} \rho dV + \int_{cs} \rho V \cdot \hat{n} \cdot dA
\]

\[
\frac{\partial}{\partial t} \int_{\text{cv}} \rho dV
\]

shows the time rate of change of the mass of the contents of the control volume.

\[
\int_{cs} \rho V \cdot \hat{n} dA
\]

shows the net rate of mass flow through the control surface.

When a flow is steady, all field properties including density remain constant with time, and the time rate of change of the mass of the contents of the control volume is zero. That is:

\[
\frac{\partial}{\partial t} \int_{\text{cv}} \rho dV = 0
\]
\[ \int \rho V \cdot \hat{n} dA \] is mass flowrate through \( dA \)

“+” is for flow out of the control volume

“-” is for flow into the control volume

It can be shown as:

\[ \int \rho V \cdot \hat{n} dA = \sum \dot{m}_{out} - \sum \dot{m}_{in} \]

An often-used expression for mass flowrate, \( \dot{m} \), through a section of control surface having area \( A \) is:

\[ \dot{m} = \rho Q = \rho AV \]

The average value of velocity, \( \bar{V} \), defined as:

\[ \bar{V} = \frac{\int \rho V \cdot \hat{n} dA}{\rho A} = V \]

If the velocity is considered uniformly distributed (one-dimensional flow) over the section \( A \), the

\[ \bar{V} = V \]
Fixed, Non-deforming Control Volume

In many applications of fluid mechanics, an appropriate control volume to use is fixed and non-deforming.

It will give the value of the time rate of change of the mass of the contents of the control volume is equal to zero.

\[
\frac{\partial}{\partial t} \int_{cv} \rho dV = 0
\]
Moving, Non-deforming Control Volume

It is sometimes necessary to use a non-deforming control volume attached to a moving reference frame.

For example, the exhaust stacks of a ship at sea and the gasoline tank of an automobile passing by.

When a moving control volume is used, the fluid velocity relative to the moving control volume (relative velocity) is an important flow field variable.

The relative velocity, $W$, is the fluid velocity seen by an observer moving with the control volume.

The control volume velocity, $V_{cv}$, is the velocity of the control volume as seen from a fixed coordinate system.

The absolute velocity, $V$, is the fluid velocity seen by a stationary observer in a fixed coordinate system.

Their relation is:

$$V = W + V_{cv}$$
We can get the control volume expression for conservation of mass (the continuity equation) for a moving, non-deforming control volume, namely:

\[
\frac{\partial}{\partial t} \int_{cv} \rho dV + \int_{cs} \rho W \cdot \hat{n} dA = 0
\]
LINEAR MOMENTUM

Derivation of the Linear Momentum Equation

Newton’s second law of motion for a system is

\[
\text{Time rate of change of the linear momentum of the system} = \text{Sum of the external forces acting on the system}
\]

Momentum is mass times velocity.

Then, the Newton’s second law becomes;

\[
\frac{D}{Dt} \int_V \rho \mathbf{v} dV = \sum F_{sys}
\]

Furthermore, for a system and the contents of a coincident control volume that is fixed and non-deforming control volume, the Reynolds transport theorem (with \( b = \text{velocity} \)) allows us to conclude that;

\[
\text{Time rate of change of the linear momentum of the system} = \text{Time rate of change of the linear momentum of the contents of the control volume} + \text{Net rate of flow of linear momentum through the control surface}
\]
It can be written as:
\[
\frac{D}{Dt} \iiint V \rho dV = \frac{\partial}{\partial t} \iiint V \rho dV + \iiint V \rho V \cdot \hat{n} dA
\]

For a control volume that is fixed and non-deforming, the appropriate mathematical statement of Newton’s second law of motion is:
\[
\frac{\partial}{\partial t} \iiint V \rho dV + \iiint V \rho V \cdot \hat{n} dA = \sum F_{\text{contents of the control volume}}
\]

We call above equation is the \textit{linear momentum equation}. 
Forces Due to Fluids in Motion

Newton’s second law of motion is

\[ F = ma \]

In fluid flow problems, we use mass flow rate (kg/s) to determine “mass” that involve in the motion.

\[ F = ma = m \cdot \frac{\Delta V}{\Delta t} = m \cdot \Delta \frac{v}{t} \]

Mass flowrate can be written as ;

\[ \dot{m} = \rho Q \]

For fluid, Newton’s second law of motion is ;

\[ F = ma = \frac{m}{\Delta t} \cdot \Delta v = \dot{m}\Delta v = \rho Q\Delta v \]
Linear momentum idea is usually used for water jet and vane problems.

Because velocities has magnitude and direction, forces act on vane can be determine as:

\[ F_x = \rho Q \Delta v_x \]
\[ F_y = \rho Q \Delta v_y \]
\[ F_R = \sqrt{F_x^2 + F_y^2} \]
Force on x-direction
\[ F_x = R_x = \rho Q \Delta v_x = \rho Q (v_{2x} - v_{1x}) = \rho Q v_1 \]

Force on y-direction
\[ F_y = R_y = \rho Q \Delta v_y = \rho Q (v_{2y} - v_{1y}) = \rho Q v_2 \]
CONTINUITY EQUATION

Derivation of the Continuity Equation

The Reynolds transport theorem states that:

\[
\frac{DB_{\text{system}}}{Dt} = \frac{D}{Dt} \int V \rho \, dV = \frac{\partial}{\partial t} \int b \rho V \, dV + \int b \rho V \hat{n} \, dA
\]

\[
B = b \cdot m = (\text{changeable parameter}) \times (\text{mass})
\]

to complete above mention Reynolds Transport Theorem, we found that, if \( B \) is mass, the \( b \) will be equal to one (1).

\[
B = \text{mass} \quad b = 1
\]

We found that the time rate of change of the system is zero, so we can say that:

\[
\frac{DB_{\text{system}}}{Dt} = \frac{Dm_{\text{system}}}{Dt} = 0
\]
\[ \frac{\partial}{\partial t} \int_{cv} \rho dV \] shows the time rate of change of the mass of the contents of the control volume.

\[ \int_{cs} \rho V \cdot \hat{n} dA \] shows the net rate of mass flow through the control surface.

When a flow is steady, all field properties including density remain constant with time, and the time rate of change of the mass of the contents of the control volume is zero. That is;

\[ \frac{\partial}{\partial t} \int_{cv} \rho dV = 0 \]
Example:

Air at standard conditions enters the compressor shown in Figure 1 at rate of 0.3 m$^3$/s. It leaves the tank through a 3cm diameter pipe with a density of 1.80 kg/m$^3$ and a uniform speed of 210 m/s.

a) Determine the rate at which the mass of air in the tank is increasing or decreasing.

b) Determine the average time rate of change of air density within the tank.
The Energy Equation

The first law of thermodynamics for a system is, in words

\[
\frac{\text{Time rate of increase of the total storage energy of the system}}{\rho g} = \frac{\text{Net time rate of energy addition by heat transfer into the system}}{2g} + \frac{\text{Net time rate of energy addition by work transfer into the system}}{2g}
\]

We could express this relation by using Bernoulli equation as :

\[
\frac{P_{\text{out}}}{\rho g} + \frac{V_{\text{out}}^2}{2g} + z_{\text{out}} = \frac{P_{\text{in}}}{\rho g} + \frac{V_{\text{in}}^2}{2g} + z_{\text{in}} + h_{\text{loss}}
\]

\[h_{\text{loss}} \text{ head loss}\]
**EXAMPLE 1**

A 30m wide river with a flowrate of 800m³/s flows over a rock pile as shown in Figure 1. Determine the direction of flow and the head loss associated with the flow across the rock pile.
**EXAMPLE 2**

An incompressible liquid flows steadily along the pipe shown in Figure 2. Determine the direction of flow and the head loss over the 6m length of pipe.
EXAMPLE 3

Water is pumped steadily through a 0.10m diameter pipe from one closed, pressurized tank to another as shown in Figure 3. The pump adds 4.0kW to the water and the head loss of the flow is 10m. Determine the velocity of the water leaving the pipe.
EXAMPLE 4

Water flows through a vertical pipe as shown in Figure 4. Is the flow up or down? Explain.
Some of the basic components of a typical pipe system are shown in Figure 1.

They include the pipes, the various fitting used to connect the individual pipes to form the desired system, the flowrate control device or valves, and the pumps or turbines that add energy to or remove energy from the fluid.
The difference between open-channel flow and the pipe flow is in the fundamental mechanism that drives the flow.

For open-channel flow, gravity alone is the driving force – the water flows down a hill.

For pipe flow, gravity may be important (the pipe need not be horizontal), but the main driving force is likely to be a pressure gradient along the pipe.

If the pipe is not full, it is not possible to maintain this pressure difference, $P_1 - P_2$. 
Laminar and Turbulent Flow

The flow of a fluid in a pipe may be laminar flow or it may be turbulent flow.

Osborne Reynolds (1842-1912), a British scientist and mathematician, was the first to distinguish the difference between these two classifications of flow by experimental.

For “small enough flowrates” the dye streak (a streakline) will remain as a well-defined line as it flows along, with only slight blurring due to molecular diffusion of the dye into the surrounding water. It is called **laminar** flow.
For “intermediate flowrate” the dye streak fluctuates in time and space, and intermittent burst of irregular behavior appear along the streak. It is called **transitional** flow.

For “large enough flowrates” the dye streak almost immediately becomes blurred and spreads across the entire pipe in a random fashion. It is called **turbulent** flow.

To determine the flow conditions, one dimensionless parameter was introduced by Osborne Reynolds, called the **Reynolds number**, $Re$.

Reynolds number, $Re$ is the ratio of the inertia to viscous effects in the flow. It is written as:

$$Re = \frac{\rho V D}{\mu}$$

- $\rho$ : Density (kg/m$^3$) 
- $V$ : Average velocity in pipe (m/s) 
- $D$ : Diameter of pipe (m) 
- $\mu$ : Dynamic viscosity (Ns/m$^2$)

The distinction between laminar and turbulent pipe flow and its dependence on an appropriate dimensionless quantity was first appointed out by Osborne Reynolds in 1883.
The Reynolds number ranges for which laminar, transitional or turbulent pipe flows are obtained cannot be precisely given.

The actual transition from laminar to turbulent flow may take place at various Reynolds numbers, depending on how much the flow is disturbed by vibration of pipe, roughness of the entrance region and other factors.

For general engineering purpose, the following values are appropriate:

1. The flow in a round pipe is laminar if the Reynolds number is less than approximately 2100.
2. The flow in a round pipe is turbulent if the Reynolds numbers is greater that approximately 4000.
3. For Reynolds number between these two limits, the flow may switch between laminar and turbulent conditions in an apparently random fashion.
The region of flow near where the fluid enters the pipe is termed the entrance region.

The shape of velocity profile in the pipe depends on whether the flow is laminar or turbulent, as does the length of the entrance region, $\ell_e$. 

Figure 4
The dimensionless entrance length, \( \frac{l_e}{D} \), correlates quite well with the Reynolds number. Typical entrance lengths are given by:

\[
\frac{l_e}{D} = 0.06(Re) \quad \text{(for laminar flow)}
\]

and

\[
\frac{l_e}{D} = 4.4(Re)^{\frac{1}{6}} \quad \text{(for turbulent flow)}
\]

for \( Re=2000 \), \( l_e = 120D \)

for \( 10^4 < Re < 10^5 \), \( 20D < l_e < 30D \)

From figure 4, the flow between (2) and (3) is termed **fully developed flow**.
Fully Developed Turbulent Flow

Turbulent in pipe flow is actually more likely to occur than laminar flow in practical situation.

Turbulent flow is a very complex process.
For pipe flow the value of the Reynolds number must be greater than approximately 4000 for turbulent flow.

For flow along a flat plate the transition between laminar and turbulent flow occurs at a Reynolds number of approximately 500,000.

\[ \text{Re}_{\text{flat plate}} = \frac{\rho VL}{\mu} \]
Losses in Pipe

It is often necessary to determine the head loss, $h_L$, that occur in a pipe flow so that the energy equation, can be used in the analysis of pipe flow problems.

The overall head loss for the pipe system consists of the head loss due to viscous effects in the straight pipes, termed the major loss and denoted $h_{L\text{-major}}$.

The head loss in various pipe components, termed the minor loss and denoted $h_{L\text{-minor}}$.

That is;

$$h_L = h_{L\text{-major}} + h_{L\text{-minor}}$$

The head loss designations of “major” and “minor” do not necessarily reflect the relative importance of each type of loss.

For a pipe system that contains many components and a relatively short length of pipe, the minor loss may actually be larger than the major loss.
Major Losses

The head loss, $h_{L\text{-major}}$ is given as:

$$h_{L\text{-major}} = f \frac{\ell V^2}{D 2g}$$

where $f$ is friction factor.

Above mention equation is called the *Darcy-Weisbach* equation. It is valid for any fully developed, steady, incompressible pipe flow, whether the pipe is horizontal or on hill.

Friction factor for laminar flow is:

$$f = \frac{64}{Re}$$

Friction factor for turbulent flow is based on *Moody* chart.

It is because, in turbulent flow, Reynolds number and relative roughness influence the friction.

Reynolds number, $Re = \frac{\rho V D}{\mu}$

Relative roughness $= \frac{\varepsilon}{D}$

(relative roughness is not present in the laminar flow)
Chapter 8 – Pipe Flow

The diagram illustrates the relationship between the friction factor $f$ and the Reynolds number $Re = \frac{\rho V D}{\mu}$ for different pipe roughness levels. The graph covers a range of $f$ values from 0.0001 to 0.1 and $Re$ values from $10^3$ to $10^7$. The chart highlights the transition from laminar to turbulent flow, with distinct regions for smooth and transition ranges. The graph also includes labels for wholly turbulent flow conditions.
The *Moody* chart is universally valid for all steady, fully developed, incompressible pipe flows.

The following equation from *Colebrook* is valid for the entire non-laminar range of the *Moody* chart. It is called *Colebrook formula*.

\[
\frac{1}{f} = -2.0 \log \left( \frac{\epsilon}{D} + \frac{2.51}{3.7 \sqrt{\text{Re}} f} \right)
\]
Minor Losses

The additional components such as valves and bend add to the overall head loss of the system, which in turn alters the losses associated with the flow through the valves.

Minor losses termed as:

\[ h_{L-minor} = K_L \frac{V^2}{2g} \]

where \( K_L \) is the loss coefficient.

Each geometry of pipe entrance has an associated loss coefficient.
Entrance flow conditions and loss coefficient.

Condition: \( \frac{A_1}{A_2} = 0 \) or \( \frac{A_1}{A_2} = \infty \)
Exit flow conditions and loss coefficient.

Condition: \[ \frac{A_1}{A_2} = 0 \quad \text{or} \quad \frac{A_1}{A_2} = \infty \]
Losses also occur because of a change in pipe diameter

For sudden contraction:

\[ h_L = K_L \frac{V^2}{2g} \]

\[ K_L = \left(1 - \frac{A_2}{A_1}\right)^2 = \left(1 - \frac{A_2}{A_c}\right)^2 = \left(1 - \frac{1}{C_c}\right)^2 \]
For sudden expansion

\[ h_L = K_L \frac{V_1^2}{2g} \]

\[ K_L = \left(1 - \frac{A_1}{A_2}\right)^2 \]
EXAMPLE 1

Water flows from the nozzle attached to the spray tank shown in Figure 1. Determine the flowrate if the loss coefficient for the nozzle (based on upstream conditions) is 0.75 and the friction factor for the rough hose is 0.11.
EXAMPLE 2

Figure 2

Water at 10 degree Celsius is pumped from a lake shown in Figure 2. If flowrate is 0.011 m³/s, what is the maximum length inlet pipe, ℓ, that can be used without cavitations occurring.
EXAMPLE 3

Water flows steadily through the 2.5cm diameter galvanized iron pipe system shown in Figure 3 at rate $6 \times 10^{-4} \text{ m}^3/\text{s}$. Your boss suggests that friction losses in the straight pipe sections are negligible compared to losses in the threaded elbows and fittings of the system. Do you agree or disagree with your boss? Support your answer with appropriate calculations.
## Table 8.1
Equivalent Roughness for New Pipes [From Moody (Ref. 7) and Colebrook (Ref. 8)]

<table>
<thead>
<tr>
<th>Pipe</th>
<th>Feet</th>
<th>Millimeters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Riveted steel</td>
<td>0.003–0.03</td>
<td>0.9–9.0</td>
</tr>
<tr>
<td>Concrete</td>
<td>0.001–0.01</td>
<td>0.3–3.0</td>
</tr>
<tr>
<td>Wood stave</td>
<td>0.0006–0.003</td>
<td>0.18–0.9</td>
</tr>
<tr>
<td>Cast iron</td>
<td>0.00085</td>
<td>0.26</td>
</tr>
<tr>
<td>Galvanized iron</td>
<td>0.0005</td>
<td>0.15</td>
</tr>
<tr>
<td>Commercial steel</td>
<td></td>
<td></td>
</tr>
<tr>
<td>or wrought iron</td>
<td>0.00015</td>
<td>0.045</td>
</tr>
<tr>
<td>Drawn tubing</td>
<td>0.000005</td>
<td>0.0015</td>
</tr>
<tr>
<td>Plastic, glass</td>
<td>0.0 (smooth)</td>
<td>0.0 (smooth)</td>
</tr>
</tbody>
</table>
A pipe system which has only one pipe line is called single pipe system.

In many pipe systems there is more than one pipe line involved, and this mechanism is called multiple pipe systems.

Multiple pipe systems are the same as for the single pipe system; however, because of the numerous unknowns involved, additional complexities may arise in solving for the flow in these systems.
Series Pipe Systems

One of the simplest multiple pipe systems is that containing pipes in series, as shown in Figure 1.

Every fluid particle that passes through the system passes through each of the pipes.

Thus, the flowrate (but not the velocity) is the same in each pipe, and the head loss from point A to point B is the sum of the head losses in each of the pipes.

The governing equations can be written as follows ;

\[ Q_1 = Q_2 = Q_3 \]

and

\[ h_{L(A-B)} = h_{L1} + h_{L2} + h_{L3} \]
Parallel Pipe Systems

Another common multiple pipe system contains pipes in parallel, as shown in Figure 2.

In this system a fluid particle traveling from A to B may take any of paths available, with the total flowrate equal to the sum of the flowrates in each pipe.

The head loss experienced by any fluid particle traveling between these two locations is independent of the path taken.

The governing equation for parallel pipe systems are;

\[ Q = Q_1 + Q_2 + Q_3 \]

and

\[ h_{L1} = h_{L2} = h_{L3} \]
Another type of pipe system called a loop is shown in Figure 3.

In this case, the flowrate through pipe (1) equals the sum of the flowrates through pipes (2) and (3), or

$$Q_1 = Q_2 + Q_3$$

The energy equation between the surfaces of each reservoir, the head loss for pipe (2) must equal that for pipe (3), even though the pipe sizes and flowrates may be different for each.
For a fluid particle traveling through pipe (1) and (2):

\[
\frac{P_A}{\rho g} + \frac{V_A^2}{2g} + z_A = \frac{P_B}{\rho g} + \frac{V_B^2}{2g} + z_B + h_{L1} + h_{L2}
\]

For a fluid particle traveling through pipe (1) and (3):

\[
\frac{P_A}{\rho g} + \frac{V_A^2}{2g} + z_A = \frac{P_B}{\rho g} + \frac{V_B^2}{2g} + z_B + h_{L1} + h_{L3}
\]

These can be combined to give:

\[
h_{L2} = h_{L3}
\]
Three-reservoir system

The branching system termed the three-reservoir problem shown in Figure 4.

With all valves open, however, it is not necessarily obvious which direction the fluid flows.

According to Figure 4, it is clear that fluid flows from reservoir A because the other reservoir levels are lower.
EXAMPLE 1

Air flows in a 0.5m diameter pipe at a rate of 10m$^3$/s. The pipe diameter changes to 0.75m through a sudden expansion. Determine the pressure rise across this expansion. Explain how there can be a pressure rise across the expansion when there is an energy loss ($K_L > 0$)

EXAMPLE 2

The surface elevations of reservoir $A$, $B$ and $C$ are at 640m, 545m and 580m respectively. A 0.45m diameter pipe, 3.2km long runs from reservoir $A$ to a node at an elevation of 610m, where the 0.3m diameter pipes each of 1.6km length branch from the original pipe and connect reservoirs $B$ and $C$. If the friction factor for each pipe is 0.020, determine the flowrate in each pipe.
EXAMPLE 3

The three water-filled tanks shown in Figure 5 are connected by pipes as indicated. If minor losses are neglected, determine the flowrate in each pipe.
BUCKINGHAM PI THEOREM

Dimensional Analysis

It is used to determine the equation is right or wrong.

The calculation is depends on the unit or dimensional conditions of the equations.

For example;

\[ F = ma \]
\[ F = MLT^{-2} \]
Unit : \( F = \text{kg.m/s} \)
**Buckingham Pi Theorem**

If an equation involving \( k \) variables is dimensionally homogeneous, it can be reduced to a relationship among \( k - r \) independent dimensionless products, where \( r \) is the minimum number of reference dimensions required to describe the variables.

For example, the function of \( G \) can be written as;

\[
G(\pi_1, \pi_2, \pi_3, \ldots, \pi_{k-r}) = 0
\]

Or

\[
\pi_1 = G(\pi_2, \pi_3, \pi_4, \ldots, \pi_{k-r}) = 0
\]

The dimensionless products are frequently referred to as “pi terms”, and the theorem is called the Buckingham Pi Theorem.

Buckingham used the symbol \( \Pi \) to represent a dimensionless product, and this notation is commonly used.
To summarize, the steps to be followed in performing a dimensional analysis using the method of repeating variables are as follows:

Step 1 List all the variables that are involved in the problem.
Step 2 Express each of the variables in terms of basic dimensions.
Step 3 Determine the required number of pi terms.
Step 4 Select a number of repeating variables, where the number required is equal to the number of reference dimensions. (usually the same as the number of basic dimensions)
Step 5 Form the pi term by multiplying one of the nonrepeating variables by the product of repeating variables each raised to an exponent that will make the combination dimensionless.
Step 6 Repeat step 5 for each of the remaining nonrepeating variables.
Step 7 Check all the resulting pi terms to make sure they are dimensionless.
Step 8 Express the final form as a relationship among the pi terms and think about what it means.
Selection of Variables

One of the most important, and difficult, steps in applying dimensional analysis to any given problem is the selection of the variables that are involved.

For most engineering problems (including areas outside fluid mechanics), pertinent variables can be classified into three groups – geometry, material properties and external effects.

Geometry:
The geometry characteristics can be usually be described by a series of lengths and angles.
Example: length [L]

Material properties:
More relates to the kinematic properties of fluid particles.
Example: velocity [LT^{-1}]

External effects:
This terminology is used to denote any variable that produces, or tends to produce, a change in the system. For fluid mechanics, variables in this class would be related to pressure, velocities, or gravity.
(combination of geometry and material properties)
Example: force [MLT^{-2}]
Common Dimensionless Groups in Fluid Mechanics

### TABLE 7.1
Some Common Variables and Dimensionless Groups in Fluid Mechanics

<table>
<thead>
<tr>
<th>Dimensionless Groups</th>
<th>Name</th>
<th>Interpretation (Index of Force Ratio Indicated)</th>
<th>Types of Applications</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \rho V \ell )</td>
<td>Reynolds number, Re</td>
<td>inertia force / viscous force</td>
<td>Generally of importance in all types of fluid dynamics problems</td>
</tr>
<tr>
<td>( \mu )</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( V )</td>
<td>Froude number, Fr</td>
<td>inertia force / gravitational force</td>
<td>Flow with a free surface</td>
</tr>
<tr>
<td>( \sqrt{g \ell} )</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( \frac{p}{\rho V^2} )</td>
<td>Euler number, Eu</td>
<td>pressure force / inertia force</td>
<td>Problems in which pressure, or pressure differences, are of interest</td>
</tr>
<tr>
<td>( \frac{\rho V^2}{E_v} )</td>
<td>Cauchy number, Ca</td>
<td>inertia force / compressibility force</td>
<td>Flows in which the compressibility of the fluid is important</td>
</tr>
<tr>
<td>( V )</td>
<td>Mach number, Ma</td>
<td>inertia force / compressibility force</td>
<td>Flows in which the compressibility of the fluid is important</td>
</tr>
<tr>
<td>( \omega \ell )</td>
<td>Strouhal number, St</td>
<td>inertia (local) force / inertia (convective) force</td>
<td>Unsteady flow with a characteristic frequency of oscillation</td>
</tr>
<tr>
<td>( \frac{\rho V^2 \ell}{v} )</td>
<td>Weber number, We</td>
<td>inertia force / surface tension force</td>
<td>Problems in which surface tension is important</td>
</tr>
</tbody>
</table>

*The Cauchy number and the Mach number are related and either can be used as an index of the relative effects of inertia and compressibility. See accompanying discussion.*
<table>
<thead>
<tr>
<th>items</th>
<th>Problems</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reynolds number</td>
<td>Flow in pipe.</td>
</tr>
<tr>
<td>Froude number</td>
<td>Flow of water around ship.</td>
</tr>
<tr>
<td></td>
<td>Flow through rivers or open conduits.</td>
</tr>
<tr>
<td>Euler number</td>
<td>Pressure problems.</td>
</tr>
<tr>
<td></td>
<td>Pressure difference between two points.</td>
</tr>
<tr>
<td>Cauchy number</td>
<td>Fluid compressibility.</td>
</tr>
<tr>
<td>Mach number</td>
<td>Fluid compressibility.</td>
</tr>
<tr>
<td>Strouhal number</td>
<td>Unsteady, oscillating flow.</td>
</tr>
<tr>
<td>Weber number</td>
<td>Interface between two fluid.</td>
</tr>
<tr>
<td></td>
<td>Surface tension problems.</td>
</tr>
</tbody>
</table>
Modeling and Similitude

Models are widely used in fluid mechanics.

Major engineering projects involving structures, aircraft, ships, rivers, harbor, and so on, frequently involve the use of models.

A model (engineering model) is a representation of a physical system that may be used to predict the behavior of the system in some desired respect.

The physical system for which the predictions are to be made is called the prototype.

Usually a model is smaller than the prototype. Therefore, it is more easily handled in the laboratory and less expensive to construct and operate than a large prototype.

However, if the prototype is very small, it may be advantageous to have a model that is larger than a prototype so that it can be more easily studied.
## Table 1.1
Dimensions Associated with Common Physical Quantities

<table>
<thead>
<tr>
<th></th>
<th>(FLT) System</th>
<th>(MLT) System</th>
</tr>
</thead>
<tbody>
<tr>
<td>Acceleration</td>
<td>(LT^{-2})</td>
<td>(LT^{-2})</td>
</tr>
<tr>
<td>Angle</td>
<td>(F^0L^0T^0)</td>
<td>(M^0L^0T^0)</td>
</tr>
<tr>
<td>Angular acceleration</td>
<td>(T^{-2})</td>
<td>(T^{-2})</td>
</tr>
<tr>
<td>Angular velocity</td>
<td>(T^{-1})</td>
<td>(T^{-1})</td>
</tr>
<tr>
<td>Area</td>
<td>(L^2)</td>
<td>(L^2)</td>
</tr>
<tr>
<td>Density</td>
<td>(FL^{-4}T^2)</td>
<td>(ML^{-3})</td>
</tr>
<tr>
<td>Energy</td>
<td>(FL)</td>
<td>(ML^2T^{-2})</td>
</tr>
<tr>
<td>Force</td>
<td>(F)</td>
<td>(MLT^{-2})</td>
</tr>
<tr>
<td>Frequency</td>
<td>(T^{-1})</td>
<td>(T^{-1})</td>
</tr>
<tr>
<td>Heat</td>
<td>(FL)</td>
<td>(ML^2T^{-2})</td>
</tr>
<tr>
<td>Length</td>
<td>(L)</td>
<td>(L)</td>
</tr>
<tr>
<td>Mass</td>
<td>(FL^{-1}T^2)</td>
<td>(M)</td>
</tr>
<tr>
<td>Modulus of elasticity</td>
<td>(FL^{-2})</td>
<td>(ML^{-1}T^{-2})</td>
</tr>
<tr>
<td>Moment of a force</td>
<td>(FL)</td>
<td>(ML^2T^{-2})</td>
</tr>
<tr>
<td>Moment of inertia (area)</td>
<td>(L^4)</td>
<td>(L^4)</td>
</tr>
<tr>
<td>Physical Quantity</td>
<td>$FLT$ System</td>
<td>$MLT$ System</td>
</tr>
<tr>
<td>----------------------------------</td>
<td>--------------</td>
<td>--------------</td>
</tr>
<tr>
<td>Moment of inertia (mass)</td>
<td>$FLT^2$</td>
<td>$ML^2$</td>
</tr>
<tr>
<td>Momentum</td>
<td>$FT$</td>
<td>$MLT^{-1}$</td>
</tr>
<tr>
<td>Power</td>
<td>$FLT^{-1}$</td>
<td>$ML^2T^{-3}$</td>
</tr>
<tr>
<td>Pressure</td>
<td>$FL^{-2}$</td>
<td>$ML^{-1}T^{-2}$</td>
</tr>
<tr>
<td>Specific heat</td>
<td>$L^2T^{-2}\Theta^{-1}$</td>
<td>$L^2T^{-2}\Theta^{-1}$</td>
</tr>
<tr>
<td>Specific weight</td>
<td>$FL^{-3}$</td>
<td>$ML^{-2}T^{-2}$</td>
</tr>
<tr>
<td>Strain</td>
<td>$F^0L^0T^0$</td>
<td>$M^0L^0T^0$</td>
</tr>
<tr>
<td>Stress</td>
<td>$FL^{-2}$</td>
<td>$ML^{-1}T^{-2}$</td>
</tr>
<tr>
<td>Surface tension</td>
<td>$FL^{-1}$</td>
<td>$MT^{-2}$</td>
</tr>
<tr>
<td>Temperature</td>
<td>$\Theta$</td>
<td>$\Theta$</td>
</tr>
<tr>
<td>Time</td>
<td>$T$</td>
<td>$T$</td>
</tr>
<tr>
<td>Torque</td>
<td>$FL$</td>
<td>$ML^2T^{-2}$</td>
</tr>
<tr>
<td>Velocity</td>
<td>$LT^{-1}$</td>
<td>$LT^{-1}$</td>
</tr>
<tr>
<td>Viscosity (dynamic)</td>
<td>$FL^{-2}T$</td>
<td>$ML^{-1}T^{-1}$</td>
</tr>
<tr>
<td>Viscosity (kinematic)</td>
<td>$L^2T^{-1}$</td>
<td>$L^2T^{-1}$</td>
</tr>
<tr>
<td>Volume</td>
<td>$L^3$</td>
<td>$L^3$</td>
</tr>
<tr>
<td>Work</td>
<td>$FL$</td>
<td>$ML^2T^{-2}$</td>
</tr>
</tbody>
</table>
Models are widely used in fluid mechanics.

Major engineering projects involving structures, aircrafts, ships, rivers, harbor, and so on, frequently involve the use of models.

A *model* is a representation of a physical system that may be used to predict the behavior of the system in some desire respect.

The physical system for which the predictions are to be made is called the *prototype*. 
Theory of Models

The theory of models can be readily developed by using the principles of dimensional analysis.

\[ \Pi_1 = \phi(\Pi_2, \Pi_3, \ldots, \Pi_n) \]

If above equation describes the behavior of a particular prototype, a similar relationship can be written for a model of this prototype, that is,

\[ \Pi_{1m} = \phi(\Pi_{2m}, \Pi_{3m}, \ldots, \Pi_{nm}) \]

Pi terms, without a subscript will refer to the prototype.

The subscript \( m \) will be used to designate the model variables or pi terms.

The pi terms can be developed so that \( \Pi_1 \) contains the variable that is to be predicted from observations made on the model. Therefore, if the model is designed and operated under the following conditions,

\[ \Pi_{2m} = \Pi_2, \Pi_{3m} = \Pi_3, \ldots, \Pi_{nm} = \Pi_n \quad \text{...... Eq.(1)} \]

Then with the presumption that the form of \( \phi \) is the same for model and prototype, it follows that,

\[ \Pi_1 = \Pi_{1m} \quad \text{...... Eq.(2)} \]
Equation (2) is the desired prediction equation and indicates that the measured value of $\Pi_{1m}$ obtained with the model will be equal to the corresponding $\Pi_1$ for the prototype as long as the other pi terms are equal.

The conditions specified by equation (1) provide the model design conditions, also called *similarity requirements* or *modeling laws*. 
Model scales

We will take the ratio of the model value to the prototype value as the scale.

Length scales are often specified.

For example, as 1:10 or as a \( \frac{1}{10} \) scale model.

The meaning of this specification is that the model is one-tenth the size of the prototype, and the tacit assumption is that all relevant lengths are scaled accordingly, so the model is geometrically similar to the prototype.

There are, however, other scales such as the;

Velocity scale, \( \frac{V_m}{V} \)

Density scale, \( \frac{\rho_m}{\rho} \)

Viscosity scale, \( \frac{\mu_m}{\mu} \)

And so on.
Models for which one or more similarity requirements are not satisfied are called *distorted models*.

Models for which all similarity requirements are met are called *true models*.