(I) INTRODUCTION (BOOK CHAPTER 1)

What is Fluid Mechanics?

First, what is a *fluid*?

- Three common states of matter are solid, liquid, and gas.
- A *fluid* is either a .
- If surface effects are not present, flow behaves similarly in all common fluids, whether gases or liquids.
- Formal definition of a fluid - A *fluid* is a substance which *deforms continuously* under the application of a *shear stress*.
  - Definition of **stress** - A stress is defined as a , acting on an infinitesimal surface element.
  - Stresses have **magnitude** and are associated with two **directions**, one for the stress itself, and one for the **surface** on which the stress acts.
  - There are *normal* stresses and *tangential* stresses.
  - *Pressure* is an example of a *normal* stress, and acts inward, toward the surface, and perpendicular to the surface.
  - A *shear* stress is an example of a *tangential* stress, i.e. it acts along the surface, parallel to the surface. Friction due to fluid viscosity is the primary source of shear stresses in a fluid.
  - One can construct a free body diagram of a little fluid particle to visualize both the normal and shear stresses acting on the body:

- **Fluids at rest cannot resist a shear stress**; in other words, when a shear stress is applied to a fluid at rest, the fluid will not remain at rest, but will move because of the shear stress.

Next, what is *mechanics*?

- Mechanics is essentially the application of the laws of force and motion. Conventionally, it is divided into two branches, .
-

So, putting it all together, there are two branches of fluid mechanics:

- is the study of fluids *at rest*. The main equation required for this is Newton's second law for non-accelerating bodies, i.e..
- is the study of fluids *in motion*. The main equation required for this is Newton's second law for accelerating bodies, i.e.

(II) PROPERTIES OF FLUIDS (BOOK CHAPTER 1)

A. Density, Specific Weight, Relative Density

Density (ρ) = mass per unit volume of substance = $\delta m/\delta v$;

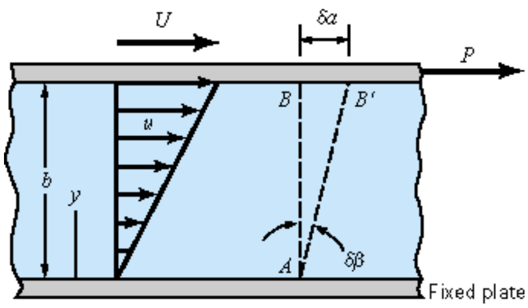
Specific weight (γ) = force exerted by the earth's gravity upon a unit volume of the substance = ρg;

Relative density (specific gravity) = ratio of mass density of the substance to that of water at a standard temperature and pressure =

B. Viscosity

Viscosity is a measure of the importance of friction in fluid flow.

Consider, for example, a fluid in two-dimensional steady shear between two parallel plates, as shown below. The bottom plate is fixed, while the upper plate is moving at a steady speed of U.



The top plate will drag the fluid along with it to the right.

Also notice that the velocity of the fluid matches that of the wall at both the top and bottom walls. This is known as the no slip condition.

It turns out (we will prove this at a later date) that the velocity profile, $u(y)$ is linear, i.e. $u(y) = Uy/b$.

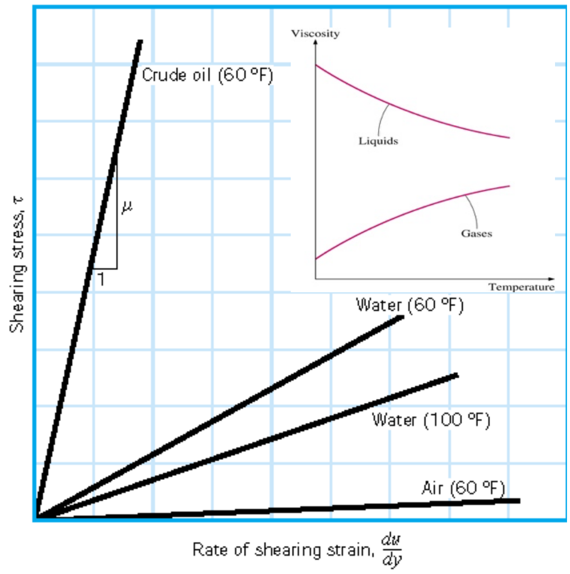
From page 1, we learnt that a definition of shear stress is given by τ = F/A. But it does not give much information for fluid properties.

To give much information about the fluid, people have given an alternative definition for shear stress

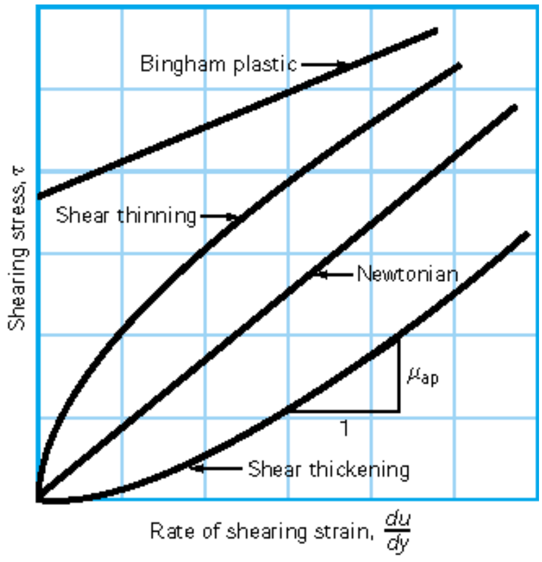
where the constant of proportionality μ (Greek letter "mu") is called **the coefficient of viscosity**. Fluids that follow the above relation are called ***Newtonian*** fluids. The coefficient of viscosity is also known as **dynamic viscosity**; its dimensions are while its SI units are .

Sometimes, it is more convenient to use **kinematic viscosity,** denoted by Greek letter "nu", which is simply defined as the viscosity divided by density, i.e.

Kinematic viscosity has the dimensions , and its SI units are .



Typically, as temperature increases, the viscosity will decrease for a liquid, but will increase for a gas.



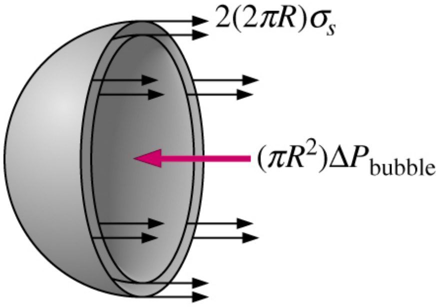
The fluid is **non-Newtonian** if the relation between shear stress and shear strain rate is **non-linear**.

C. <u>Surface Tension</u>

Surface tension is a property of liquids which is felt at the interface between the liquid and another fluid (typically a gas). Surface tension has dimensions of ,
and always acts parallel to the interface..

A soap bubble is a good example to illustrate the effects of surface tension. How does a soap bubble remain spherical in shape? The answer is that there is a higher pressure inside the bubble than outside, much like a balloon. In fact, surface tension in the soap film acts much the same as the tension in the skin of a balloon.

Consider a soap bubble of radius *R* with internal pressure p_{in} and external (atmospheric) pressure p_{out}. The excess pressure $\Delta P_{\text{bubble}} = P_{\text{in}} - P_{\text{out}}$ can be found by considering the free-body diagram of half a bubble. Note that surface tension acts along the circumference (resulting from cutting across the two interfaces) and the pressure acts on the area of the half-bubble. By statics (to be explained later), the net force due to the pressure is equal to the pressure times the projected area. Hence, balancing the forces due to surface tension and pressure difference:



(*b*) Half a bubble

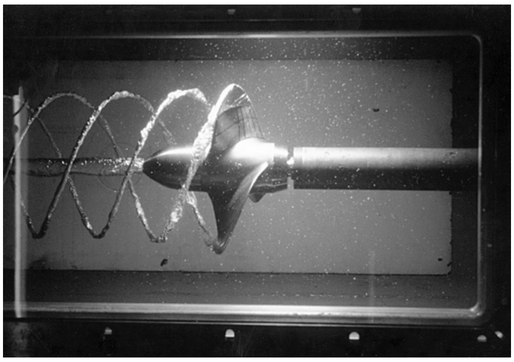
where σ_s is the surface tension of the fluid in air.

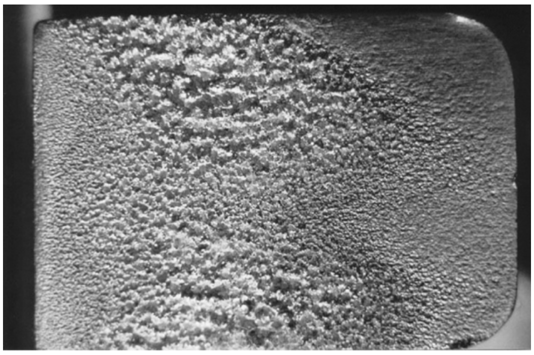
You may repeat this exercise for a droplet, and show that

D. Vapor Pressure

Vapor pressure is defined as the pressure at which a liquid will boil (vaporize). Vapor pressure rises as temperature rises. For example, suppose you are camping on a very high mountain (10,000 ft.). The atmospheric pressure at this elevation is about 70 kPa. One can find that at a temperature of around 90ºC, the vapor pressure of water is also about 70 kPa. From this it can be inferred that at 10,000 ft. of elevation, water boils at around 90ºC, rather than the common 100ºC at standard sea level pressure. This has consequences for cooking. For example, eggs have to be cooked longer at elevation to become hard-boiled since they cook at a lower temperature. A pressure cooker has the opposite effect. Namely, the tight lid on a pressure cooker causes the pressure to increase above the normal atmospheric value. This causes water to boil at a temperature even greater than 100ºC; eggs can be cooked a lot faster in a pressure cooker!

Vapor pressure is important to fluid flows because, in general, pressure in a flow decreases as velocity increases. This can lead to ***cavitation***, which is generally destructive and undesirable. In particular, at high speeds the local pressure of a liquid sometimes drops below the vapor pressure of the liquid. In such a case, ***cavitation*** occurs. In other words, a "cavity" or bubble of vapor appears because the liquid vaporizes or boils at the location where the pressure dips below the local vapor pressure. Cavitation is not desirable for several reasons. First, it causes noise (as the cavitation bubbles collapse when they migrate into regions of higher pressure). Second, it can lead to inefficiencies and reduction of heat transfer in pumps and turbines (turbomachines). Finally, the collapse of these cavitation bubbles causes pitting and corrosion of blades and other surfaces nearby. The left figure below shows a cavitating propeller in a water tunnel, and the right figure shows cavitation damage on a blade.





E. Compressibility

All fluids are compressible under the application of external forces. The compressibility of a fluid is expressed by its **bulk modulus of elasticity** *E*, which is the ratio of the change in unit pressure to the corresponding volume change per unit volume.

$$E = \frac{\Delta P}{-\Delta V / V} = \frac{\Delta P}{\Delta \rho / \rho}$$

Note that the bulk modulus of elasticity has the same dimensions as pressure: $[E] = [ML^{-1}T^{-2}]$.

For water at room temperature, E is approximately 2.2×10^9 N/m^2, while for air at atmospheric pressure the isentropic bulk modulus of elasticity is approximately 1.4×10^5 N/m^2. That is, air is typically four orders of magnitude more compressible than water.

For most practical purposes liquids may be regarded as incompressible. However, there are certain cases, such as unsteady flow in pipes (e.g., water hammer), where the compressibility should be taken into account. Gases may also be treated as incompressible if the change in density is very small (typically less than 3%).

An ideal fluid is an incompressible fluid.

Pressure disturbances imposed on a fluid move in waves. These pressure waves move at a velocity equal to that of sound through the fluid. The velocity, or celerity, c, is given by

$$c = \sqrt{E / \rho}$$

Compressibility is important to high-speed air flow when the **Mach number** (= velocity of flow/sound speed = V/c) is larger than 0.3

F. <u>Perfect Gas Law</u>

Very often we have fluid flows of gases at, or near, atmospheric pressure. In these cases, the changes in pressure p, density ρ and absolute temperature T of a gas particle may be related accurately to each other by the perfect (or ideal) gas law:

$$p = \rho RT, \qquad \text{where} \quad R = R_g / M_g$$

where R is called the perfect gas constant, R_g is the Universal gas constant and M_g is the gas molecular weight.

The universal gas constant is $R_g \cong 8.31$ J/mol· K $\cong 0.082$ L·atm/mol·K.

The perfect gas law alone is insufficient to explain how the properties of a gas change as it moves. In addition, the laws of thermodynamics must be invoked. Compressible flows are inherently complicated because the laws of thermodynamics, as well as the laws of fluid mechanics, operate simultaneously.

G. <u>Concluding Remarks</u>

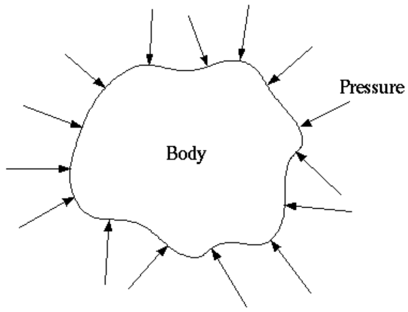
Fluid mechanics represents that branch of applied mechanics dealing with the behavior of fluids at rest and in motion. In the development of the principles of fluid mechanics, some fluid properties play principal roles, other only minor roles or no roles at all for a particular problem. In fluid statics, weight is the important property, whereas in fluid flow, density and viscosity are predominant properties. Where appreciable compressibility occurs, principles of thermodynamics must be considered. Vapor pressure becomes important when low gauge pressures are involved, and surface tension affects static and flow conditions in small passages.

(III) FLUID STATICS (BOOK CHAPTER 2)

Hydrostatics is the study of pressures throughout a fluid at rest and the pressure forces on finite surfaces. As the fluid is at rest, there are no shear stresses in it. Hence the pressure at a point on a plane surface always acts normal to the surface, and all forces are independent of viscosity. **The pressure variation is due only to the weight of the fluid**. As a result, the controlling laws are relatively simple, and analysis is based on a straightforward application of the mechanical principles of force and moment. Solutions are exact and there is no need to have recourse to experiment.

A. Introduction to Pressure

Pressure always acts **inward normal** to any surface (even imaginary surfaces as in a control volume).

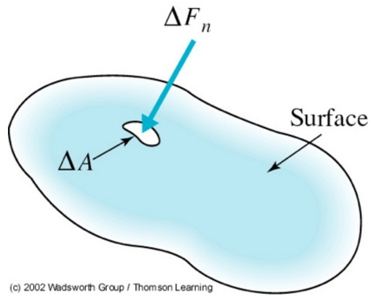


Pressure is a normal stress, and hence has dimensions of force per unit area, or [$ML^{-1}T^{-2}$]. In the English system of units, pressure is expressed as "psi" or lbf/in^2. In the Metric system of units, pressure is expressed as "pascals" (Pa) or N/m^2.

Standard atmospheric pressure is 101.3 kPa or 14.69 psi.

Pressure is formally defined to be

$$p = \lim_{\Delta A \to 0} \frac{\Delta F_n}{\Delta A}$$



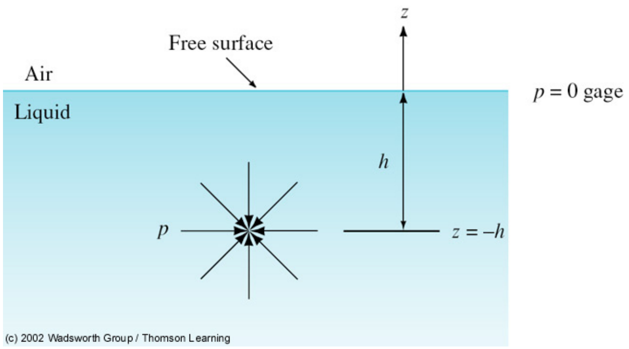
where ΔF_n is the normal compressive force acting on an infinitesimal area ΔA.

B. Pressure at a Point

By considering the equilibrium of a small triangular wedge of fluid extracted from a static fluid body, one can show that for *any* wedge angle θ, the pressures on the three faces of the wedge are equal in magnitude:

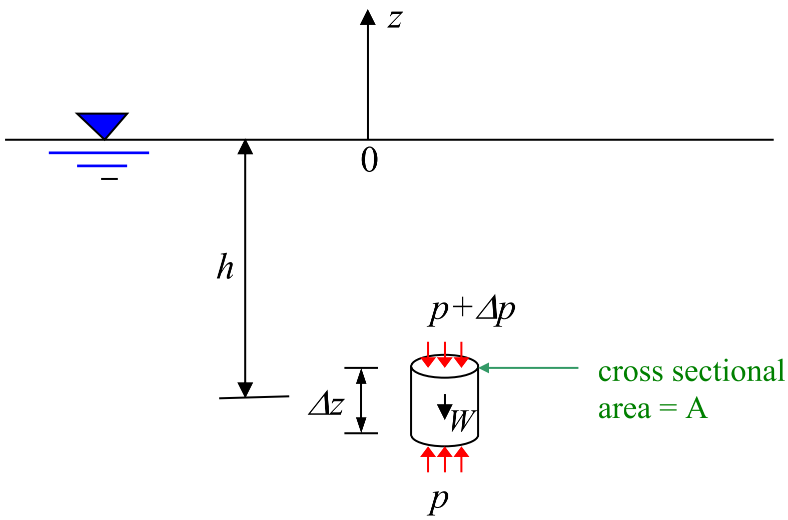
This result is known as **Pascal's law**, which states that the pressure at a point in a fluid at rest, or in motion, is independent of direction as long as there are no shear stresses present.

Proof of Pascal’s law:



Pressure at a point has the same magnitude in all directions, and is called **isotropic**.

C. <u>Pressure Variation with Depth</u>



Consider a small vertical cylinder of fluid in equilibrium, where *positive z is pointing vertically upward*. Suppose the origin $z = 0$ is set at the free surface of the fluid. Then the pressure variation at a depth $z = -h$ below the free surface is governed by

Therefore, the hydrostatic pressure increases with depth at the rate of the specific weight $\gamma \equiv \rho g$ of the fluid.

Homogeneous fluid: ρ is constant

By simply integrating the above equation:

where C is an integration constant. When z = 0 (on the free surface), $p = C = p_0$ (the atmospheric pressure). Hence, Pressure given by this equation is called ABSOLUTE PRESSURE, i.e., measured above perfect vacuum.

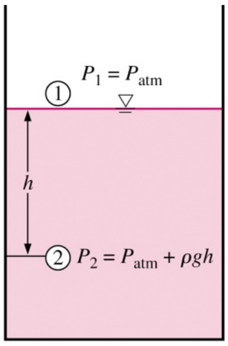
However, for engineering purposes, it is more convenient to measure the pressure above a datum pressure at atmospheric pressure. By setting $p_0 = 0$,

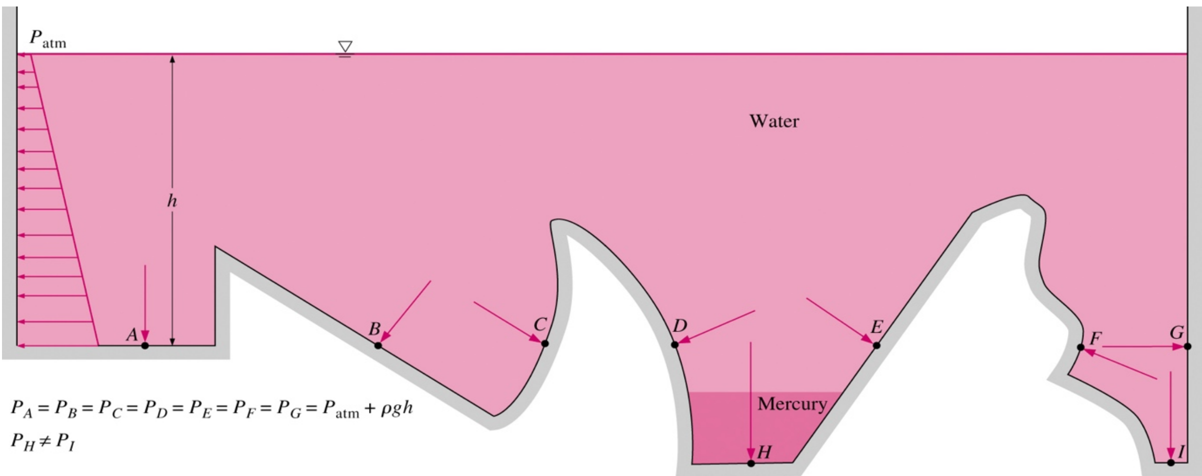
Pressure given by this equation is called GAUGE (GAGE) PRESSURE.

The equation derived above shows that when the density is constant, **the pressure in a liquid at rest increases linearly with depth from the free surface**.

Consequently, the distribution of pressure acting on a submerged flat surface is always trapezoidal (or triangular if the surface pierces through the free surface of the liquid and the pressure is gauge pressure).

Also, the pressure is the same at all points with the same depth from the free surface regardless of geometry, provided that the points are interconnected by the same fluid. However, the thrust due to pressure is perpendicular to the surface on which the pressure acts, and hence its direction depends on the geometry.





(Optional) Compressible fluid: ρ varies with depth

Example: Find the relationship between pressure and altitude in the atmosphere near the Earth's surface. For simplicity, neglect the vertical temperature gradient. Let temperature T = 288 K (15°C) and pressure p_0 = 1 atm at the surface. The average molecular weight of air is M_g = 28.8 g/mol. The Universal gas constant is R_g = 8.3 J/mol· K.

Solution: Let the altitude above the Earth's surface be denoted by z, then

$$\frac{dp}{dz} = -\rho g$$

Assume that air is a perfect gas, its density varies with pressure according to

$$\rho = P\frac{M_g}{R_g T}$$

Combining the above two equations, and integrate:

$$\frac{dp}{dz} = -p\frac{M_g g}{R_g T} \quad \Rightarrow \frac{dp}{p} = -\frac{M_g g}{R_g T}dz$$

$$\Rightarrow \int_{p_0}^{p}\frac{dp}{p} = -\int_0^z \frac{M_g g}{R_g T}dz$$

$$\Rightarrow \ln\frac{p}{p_0} = -\frac{M_g g}{R_g T}z$$

$$\Rightarrow p = p_0 \exp\left[-\left(\frac{M_g g}{R_g T}\right)z\right]$$

Neglecting temperature variation, the exponential decay rate for pressure with height is,

$$\frac{M_g g}{R_g T} = \frac{28.8\times10^{-3}\times9.81}{8.3\times288} = 1.18\times10^{-4} \text{ per meter of rise}$$

Say, at 2000 ft or 610 m above the Earth's surface, the pressure is

$$p = (1 \text{ atm})\exp\left[-1.18\times10^{-4}\times610\right] = 0.93 \text{ atm}$$

That is, for such a high elevation, the pressure drops only by 7%. (Note that temperature cannot be considered constant if this calculation is performed for large altitude differences.)

In most practical problems where the change in elevation is not extremely large, atmospheric pressure can be assumed to be constant.

D. Hydrostatic Pressure Difference Between Two Points

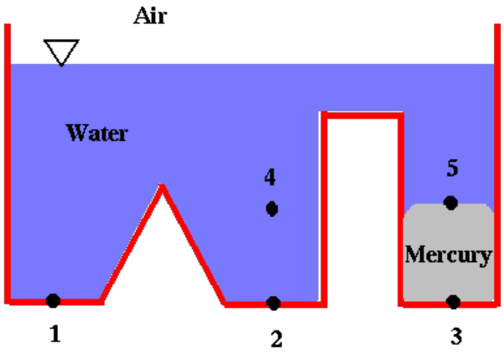
For a fluid with constant density,

It is easily remembered by thinking about scuba diving. As a diver goes down, the pressure on his ears increases. So, the pressure "below" is greater than the pressure "above."

There are several "rules" or comments which directly result from the above equation:

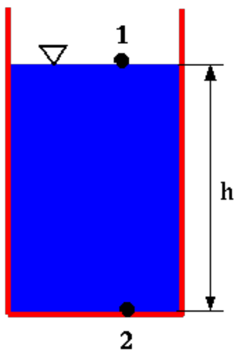
- **If you can draw a continuous line through the same fluid from point 1 to point 2, then $p_1 = p_2$ if $z_1 = z_2$.**

  For example, consider the oddly shaped container:



- **Any free surface open to the atmosphere has atmospheric pressure, p_0.**

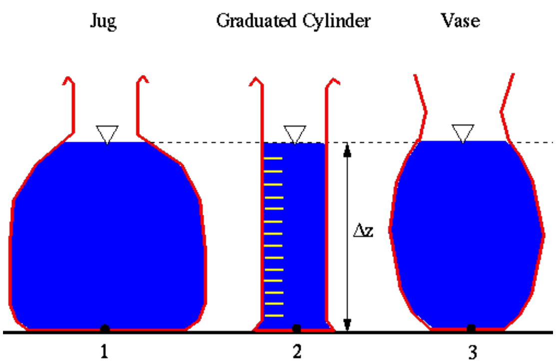
  (This rule holds not only for hydrostatics, but for any free surface exposed to the atmosphere, whether the surface is moving, stationary, flat, or mildly curved.) Consider the hydrostatics example of a container of water:



- **The shape of a container does not matter in hydrostatics.**

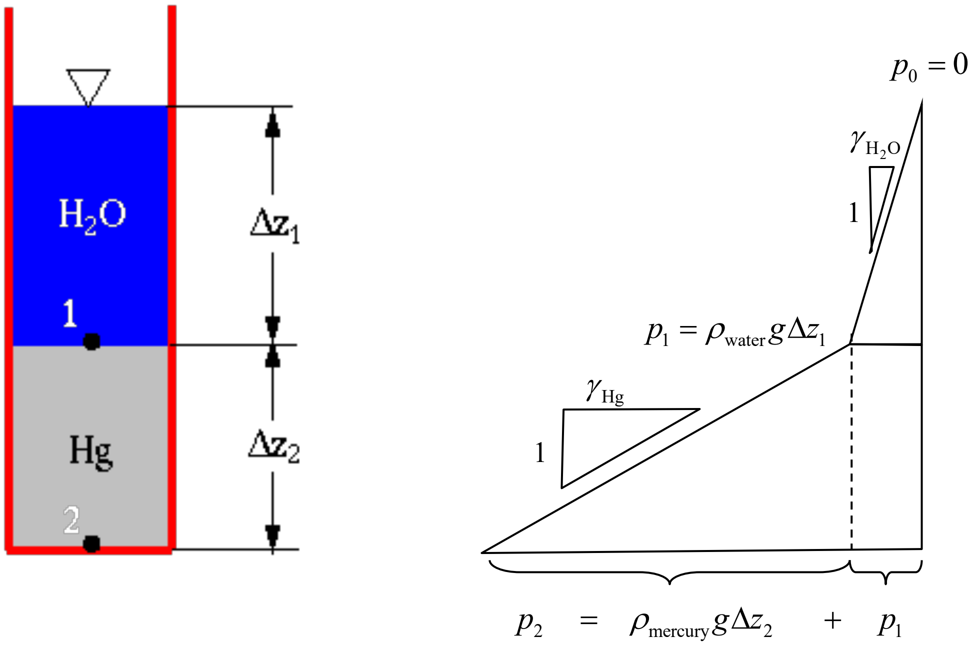
  (Except of course for very small diameter tubes, where surface tension becomes important.) Consider the three containers in the figure below:

  At first glance, it may seem that the pressure at point 3 would be greater than that at point 1 or 2, since the weight of the water is more "concentrated" on the small area at the bottom, but in reality, all three pressures are identical. Use of our hydrostatics equation confirms this conclusion, i.e.



- **Pressure in layered fluid**.

  For example, consider the container in the figure below, which is partially filled with mercury, and partially with water:

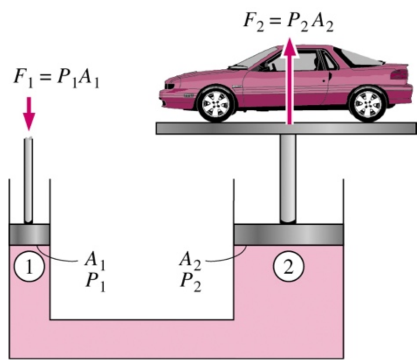


In this case, our hydrostatics equation must be used twice, once in each of the liquids

Shown on the right side of the above figure is the distribution of pressure with depth across the two layers of fluids, where the atmospheric pressure is taken to be zero $p_0 = 0$. Note that:

- The pressure is continuous at the interface between water and mercury. Therefore, p_1, which is the pressure at the bottom of the water column, is the starting pressure at the top of the mercury column. The pressure p_1 can also be regarded as the water surcharge pressure superimposed onto (uniformly transmitted to, and felt at any depth by) the mercury below.
- The vertical gradient of the pressure distribution is equal to the specific weight of the fluid γ. Therefore, the pressure in mercury increases with depth at a rate 13.6 times faster than that in water since $\gamma_{\text{mercury}} / \gamma_{\text{water}} = 13.6$.

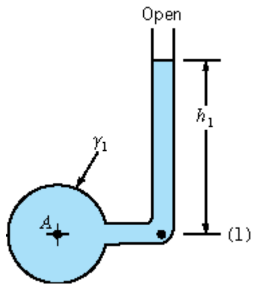
The fact that the *pressure (or known as surcharge) applied to a confined fluid increases the pressure throughout the fluid by the same amount* has important applications, such as in the hydraulic lifting of heavy objects:



$$P_1 = P_2 \Rightarrow \frac{F_1}{A_1} = \frac{F_2}{A_2} \Rightarrow \frac{F_1}{F_2} = \frac{A_1}{A_2} \ll 1$$

E. <u>Pressure Measurement and Manometers</u>

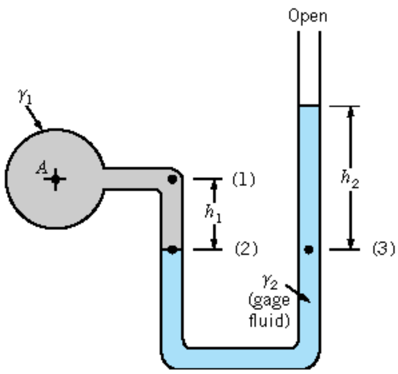
- **Piezometer tube**



The simplest manometer is a tube, open at the top, which is attached to a vessel or a pipe containing liquid at a pressure (higher than atmospheric) to be measured. This simple device is known as a piezometer tube. As the tube is open to the atmosphere the pressure measured is relative to atmospheric so is **gauge** pressure:

This method can only be used for liquids (i.e. not for gases) and only when the liquid height is convenient to measure. It must not be too small or too large and pressure changes must be detectable.

- **U-tube manometer**



This device consists of a glass tube bent into the shape of a "U", and is used to measure some unknown pressure. For example, consider a U-tube manometer that is used to measure pressure p_A in some kind of tank or machine.

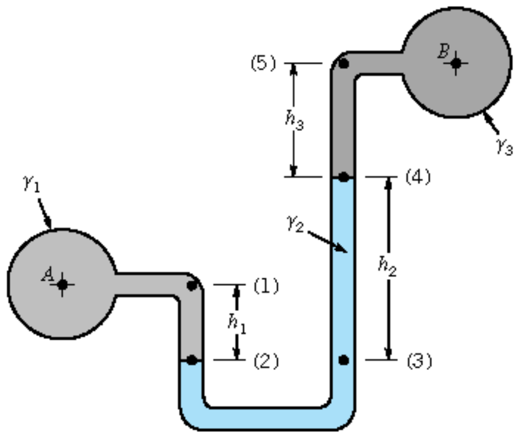
Again, the equation for hydrostatics is used to calculate the unknown pressure. Consider the left side and the right side of the manometer separately:

Since points labeled (2) and (3) in the figure are at the same elevation in the same fluid, they are at equivalent pressures, and the two equations above can be equated to give

Finally, note that in many cases (such as with air pressure being measured by a mercury manometer), the density of manometer fluid 2 is much greater than that of fluid 1. In such cases, the last term on the right is sometimes neglected.

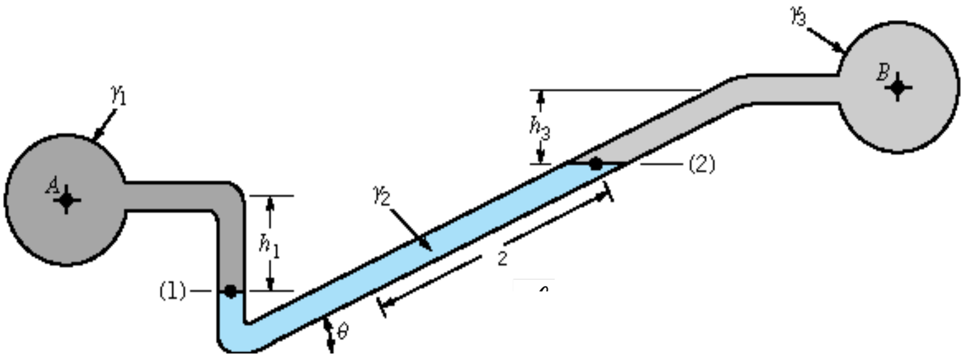
- **Differential manometer**

A differential manometer can be used to measure the difference in pressure between two containers or two points in the same system. Again, on equating the pressures at points labeled (2) and (3), we may get an expression for the pressure difference between *A* and *B*:



In the common case when *A* and *B* are at the same elevation $(h_1 = h_2 + h_3)$ and the fluids in the two containers are the same $(\gamma_1 = \gamma_3)$, one may show that the pressure difference registered by a differential manometer is given by where ρ_m is the density of the manometer fluid, ρ is the density of the fluid in the system, and h is the manometer differential reading.

- **Inclined-tube manometer**

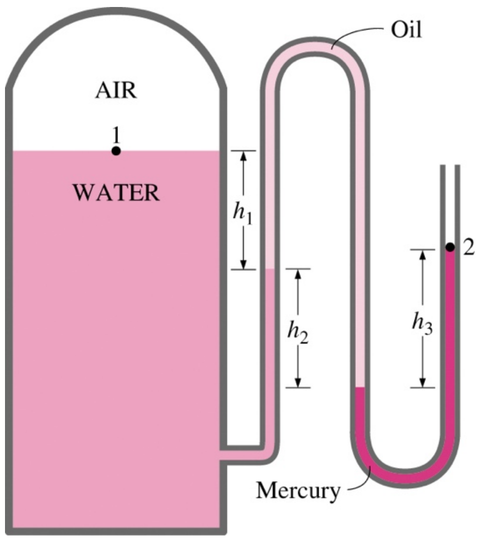


As shown above, the differential reading is proportional to the pressure difference. If the pressure difference is very small, the reading may be too small to be measured with good accuracy. To increase the sensitivity of the differential reading, one leg of the manometer can be inclined at an angle θ, and the differential reading is measured along the inclined tube. As shown above, $h_2 = \ell_2 \sin\theta$, and hence

Obviously, the smaller the angle θ, the more the reading ℓ_2 is magnified.

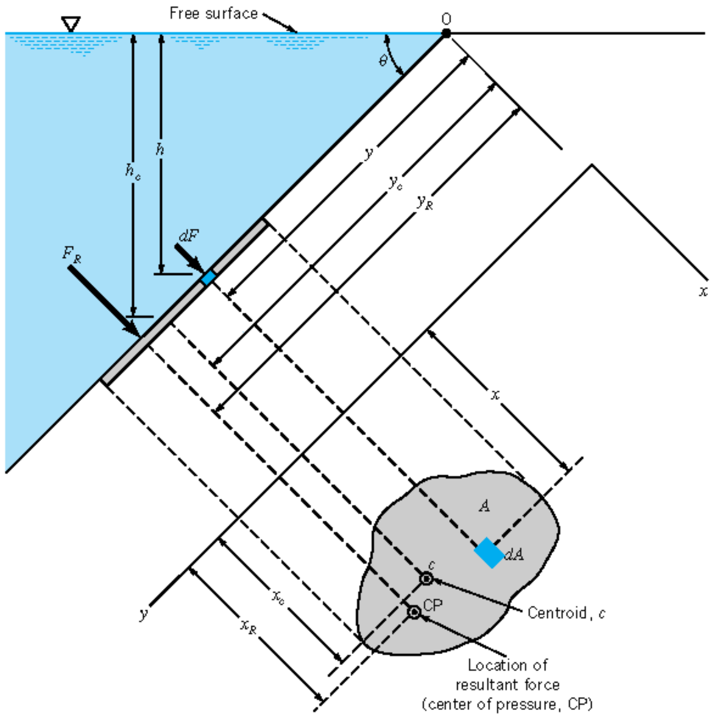
- **Multifluid manometer**

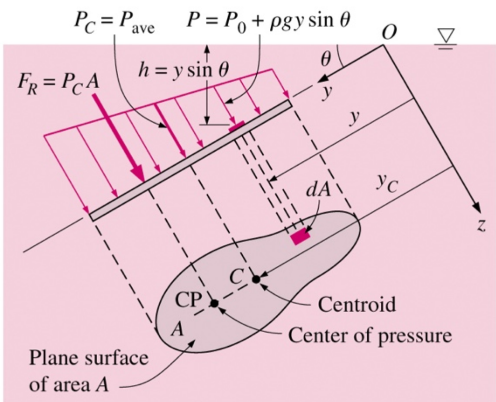
The pressure in a pressurized tank is measured by a multifluid manometer, as is shown in the figure. Show that the air pressure in the tank is given by

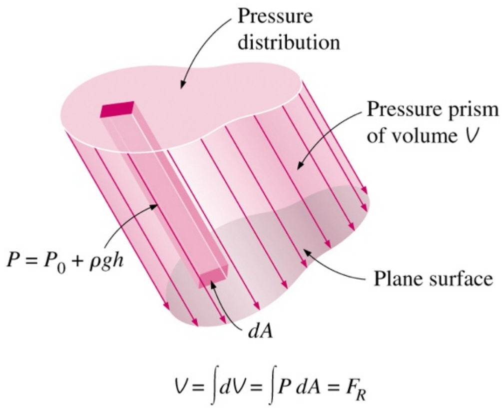


F. Hydrostatic Force on a Plane Surface

Suppose a *submerged* plane surface is inclined at an angle θ to the free surface of a liquid







Notation:-

A - area of the plane surface
O - the line where the plane in which the surface lies intersects the free surface,
C - centroid (or center of area) of the plane surface,
CP - center of pressure (point of application of the resultant force on the plane surface),
F_R - magnitude of the resultant force on the plane surface (acting normally),
h_c - vertical depth of the centroid C,
h_R - vertical depth of the center of pressure CP,
y_c - inclined distance from O to C,
y_R - inclined distance from O to CP.

Find magnitude of resultant force:

The resultant force is found by integrating the force due to hydrostatic pressure on an element dA at a depth h over the whole surface:

$$F_R = \int_A dF = \int_A \rho g h dA = \rho g \sin\theta \int_A y dA$$

where by the first moment of area

$$\int_A y dA = y_c A$$

Hence,

$$\boxed{F_R = \rho g \left(y_c \sin\theta \right) A = \rho g h_c A}$$

The resultant force on one side of any plane submerged surface in a homogeneous fluid is therefore equal to the pressure at the centroid of the surface (the mean pressure) times the area of the surface. This relationship is true irrespective of the shape of the plane or the angle θ at which it is slanted.

Find location of center of pressure:

Taking moment about O,

$$F_R y_R = \int_A y dF \Rightarrow \left(\rho g y_c \sin\theta A \right) y_R = \int_A y \left(\rho g y \sin\theta dA \right) \Rightarrow \left(y_c A \right) y_R = \int_A y^2 dA$$

But

$$\int_A y^2 dA = I_O = I_c + A y_c^2 \quad \text{by parallel axis theorem}$$

where I_O = second moment of area (or moment of inertia) of the surface about O,

I_c = second moment of area (or moment of inertia) about an axis through the centroid and parallel to the axis through O (depends on the geometry of the surface, see below for the values for some common figures).

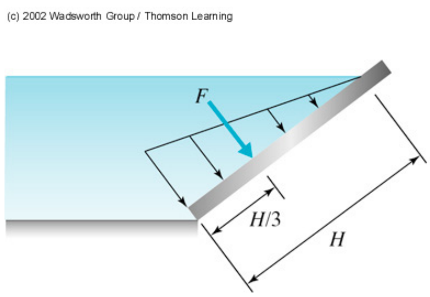
Therefore, on substituting,

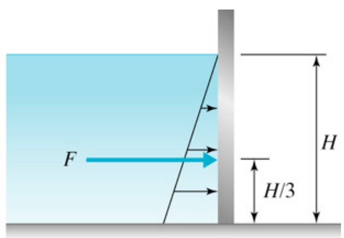
$$\left(y_c A \right) y_R = A y_c^2 + I_c$$

$$\Rightarrow \boxed{y_R = y_c + \frac{I_c}{y_c A}} \qquad \text{or} \qquad \boxed{h_R = h_c + \frac{I_c \sin^2\theta}{h_c A}}$$

Now, the depth of the center of pressure depends on the shape of the surface and the angle of inclination, and is always below the depth of the centroid of the plane surface for $\theta \neq 0$.

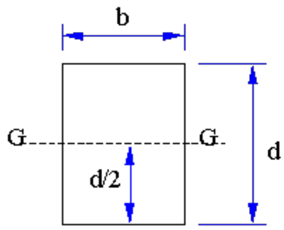
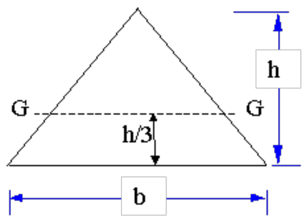
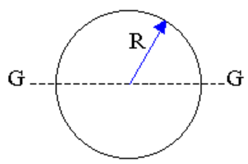
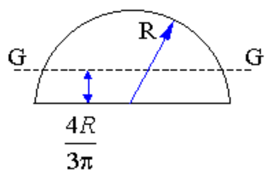
For a flat rectangular surface that pierces through the free surface, and hence a triangular pressure distribution: $A = HB, \quad h_c = \frac{1}{2} H \sin\theta, \quad h_R = \frac{2}{3} H \sin\theta, \quad F = \frac{1}{2} \rho g H^2 B \sin\theta$





Properties for some Common sectional areas

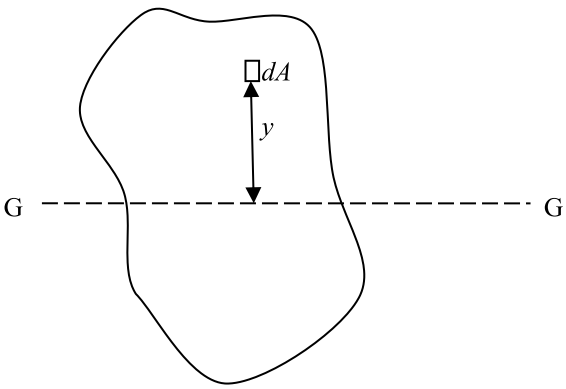
GG is an axis passing through the centroid and parallel to the base of the figure.

Shape	Dimensions	Area	Second moment of area about GG I_C
Rectangle		bd	$\frac{bd^3}{12}$
Triangle		$\frac{bh}{2}$	$\frac{bh^3}{36}$
Circle		πR^2	$\frac{\pi R^4}{4}$
Semi-Circle		$\frac{\pi R^2}{2}$	$0.11R^4$

Some Additional Notes on Second Moment of Area

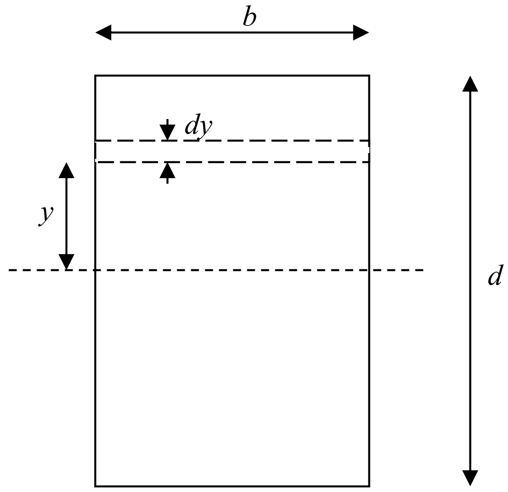
For a plane surface of arbitrary shape, we may define the n^{th} (n = 0, 1, 2, 3, ...) moment of area about an axis GG by the integral

$$\int_A y^n dA ,$$



Then,

- the zeroth moment of area = total area of the surface,
- the first moment of area = 0, if GG passes through the centroid of the surface,
- the second moment of area gives the variance of the distribution of area about the axis.

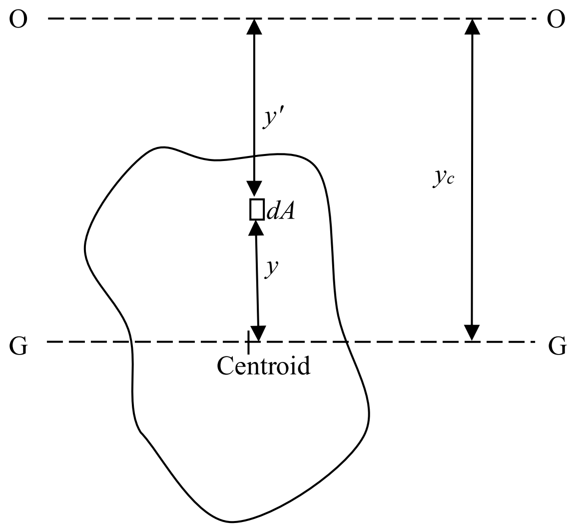


For example, for a rectangular surface, the second moment of area about the axis that passes through the centroid is

$$\begin{aligned} I_c &= \int_A y^2 dA \\ &= \int_{-d/2}^{d/2} y^2 \left(bdy\right) \\ &= \left[\frac{by^3}{3}\right]_{-d/2}^{d/2} \\ &= \frac{bd^3}{12} \end{aligned}$$

Parallel Axis Theorem

If OO is an axis that is parallel to the axis GG, which passes through the centroid of the surface, then the second moment of area about OO is equal to that about GG plus the square of the distance between the two axes times the total area:

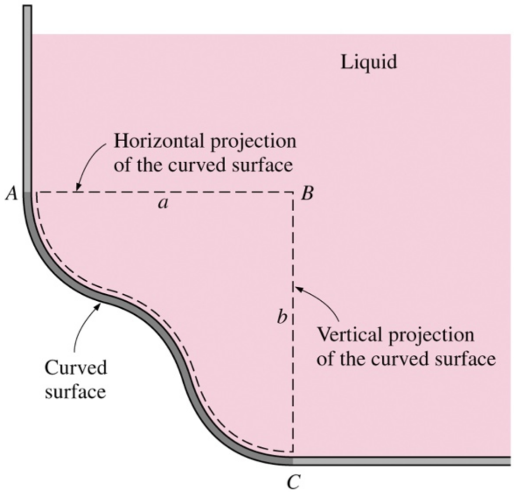


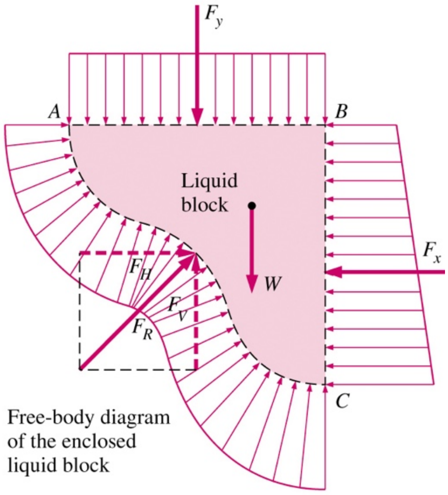
$$\begin{aligned} I_o &= \int_A y'^2 dA \\ &= \int_A \left(y_c - y\right)^2 dA \\ &= \int_A \left(y_c^2 - 2y_c y + y^2\right) dA \\ &= y_c^2 A - 2y_c \underbrace{\int_A y dA}_{0} + \underbrace{\int_A y^2 dA}_{I_c} \\ &= y_c^2 A + I_c \end{aligned}$$

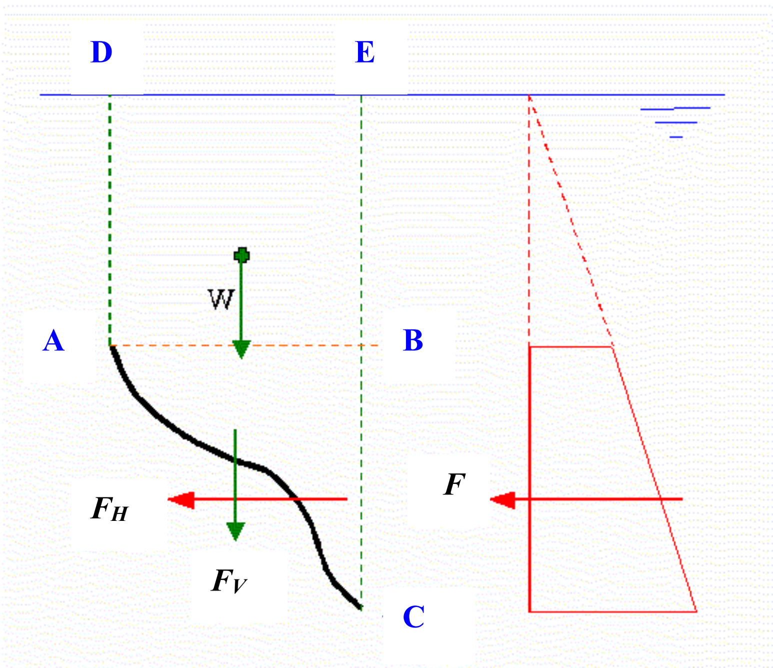
G. <u>Hydrostatic Force on Submerged Curved Surfaces</u>

1) Liquid above surface

Suppose we are required to find the force acting on the upper side of the curved surface AC.







Horizontal component of force on surface:

By considering the equilibrium of the liquid mass contained in ABC, we get

$F_H = F$ = resultant force of liquid acting on the projection of the curved surface on a vertical plane (BC) and acting through the center of pressure of F.

Vertical component of force on surface

By considering the equilibrium of the liquid mass contained in ADEC, we get

$F_V = W$ = weight of liquid vertically above the surface (ADEC) and through the center of gravity of the liquid mass.

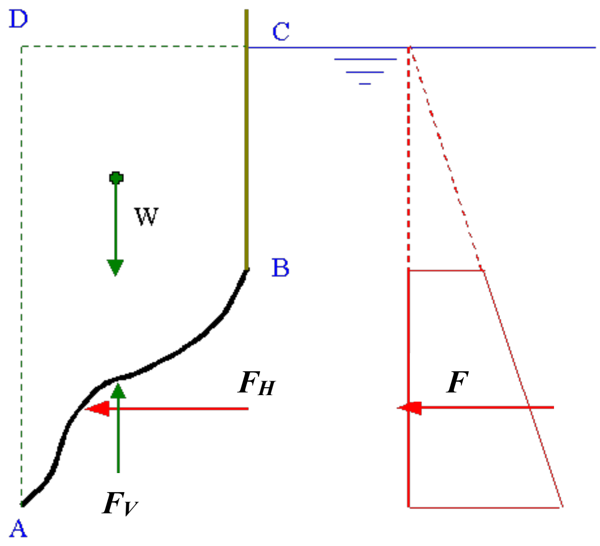
Resultant force

$$F_R = \sqrt{F_H^2 + F_V^2} \ ,$$

pointing **downward**, and making an angle $\alpha = \tan^{-1}(F_V / F_H)$ with the horizontal.

2) Liquid below surface

Suppose we are required to find the force acting on the underside of the curved surface AB. The space above the surface ADCB may be empty or contain other fluid.



Imagine that the space (ADCB) vertically above the curved surface is occupied with the same fluid as that below it (disregard what actually is filling that space). Then the surface AB could be removed without disrupting the equilibrium of the fluid. That means, the force acting on the underside of the surface would be balanced by that acting on the upper side under this imaginary condition. Therefore we may use the same arguments as in the preceding case:

Horizontal component of force on surface:

$F_H = F$ = resultant force of liquid acting on the projected vertical area (AB) and acting through the center of pressure of F.

Vertical component of force on surface

$F_V = W$ = weight of imaginary liquid (i.e., same liquid as on the other side of the surface) vertically above the surface (ADCB) and through the center of gravity of the liquid mass.

Resultant force

$$F_R = \sqrt{F_H^2 + F_V^2} \ ,$$

which points **upward**, and makes an angle $\alpha = \tan^{-1}(F_V / F_H)$ with the horizontal.

H. Buoyancy

Because the pressure in a fluid in equilibrium increases with depth, the fluid exerts a resultant upward force on any body which is wholly or partly immersed in it. This force is known as the **upthrust** or **buoyant force**, which can be determined according to Archimedes' principle or the law of flotation.

The buoyant force acting on a body immersed in a fluid is equal to the weight of the fluid displaced by the body, and it acts upward through the centroid of the displaced volume.

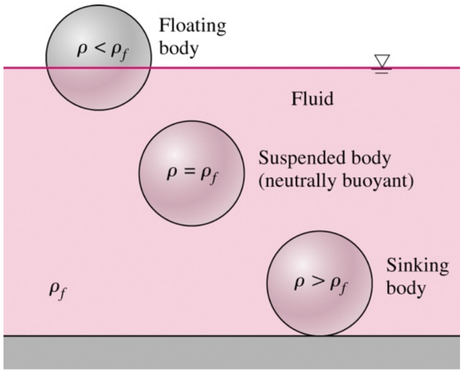
Note that F_B is positive. Previous chapters do not involve the determination of F_B, because they concern an immersed object where fluid only acts on one side of the object. For instance, we computed the hydrostatic force acting on the wall of a container filled with liquid, i.e. there is no liquid outside of the container. In contrast, we need to consider F_B if the object is immersed in the liquid, i.e. there is liquid on both sides of the object, for example a fish swimming under water.

- One way to show this law of flotation is to consider the free-body diagram of an immersed object. The weight of the object, W, is acting downward whereas the buoyancy force, F_B, is acting upward. The sum of W and F_B is the net force acting on the object, which defines the apparent weight of the object in a fluid:

- W_{app} of an immersed object is always smaller than its actual weight W, because of a positive F_B. Equivalently, W_{app} is the net force on an object

- A body will *sink* in a liquid when its weight is larger than the weight of the fluid displaced by the body, i.e the net force of the body, or equivalently W_{app}, is greater than zero. This amounts to the condition that the density of the body is larger than the density of the fluid, or the body is denser than the fluid. Apparent weight is

- A *neutrally buoyant* body is one with the same density as the fluid; it can be suspended anywhere in the fluid. Apparent weight W_{app} (net force) is zero.
- A body will *float* in a liquid when its density is less than the density of the fluid, or the body is lighter than the fluid. The body can only displace as much as its own weight of fluid when freely floating. Apparent weight is zero (The "smallest" W_{app} is zero, which is for a floating object, and is the same for a neutrally buoyant object).



Computing hydrostatic force and its location

1 Computing magnitude of hydrostatic force, F_R

To summarize what we have learnt so far,

- For a fluid at rest, the force acting on it must be perpendicular to its surface because there are no shearing stresses present.
- The pressure will vary linearly with depth, $P = \rho g h$.

Next, we are going to apply what we have learnt about hydrostatics to compute the force acting on a submerged object in a hydrostatic system.

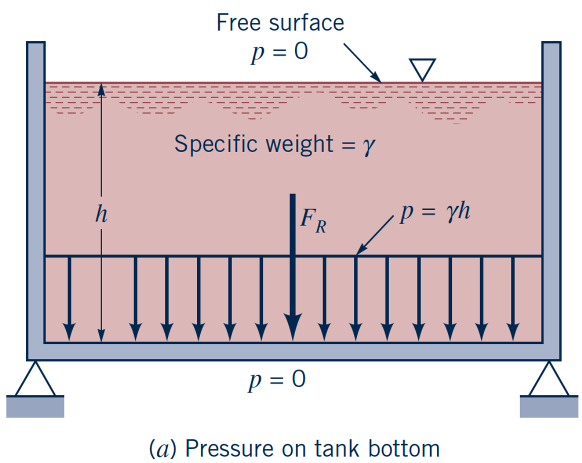


FIG. 1

- Refer to Fig. 1(a): Computing the resultant force at bottom is simple, $F_R = PA$, where A is the area of the bottom.
- Refer to Fig. 1(b): Computing the resultant force on the sides are more complicated for the following reasons: (i) Pressure varies with depth. (ii) In general, the sides may be non-rectangular shape. (iii) The sides may be inclined.

To consider a general case, let's consider an inclined object of arbitrary shape submerged in a liquid, as shown in Fig. 2 below.

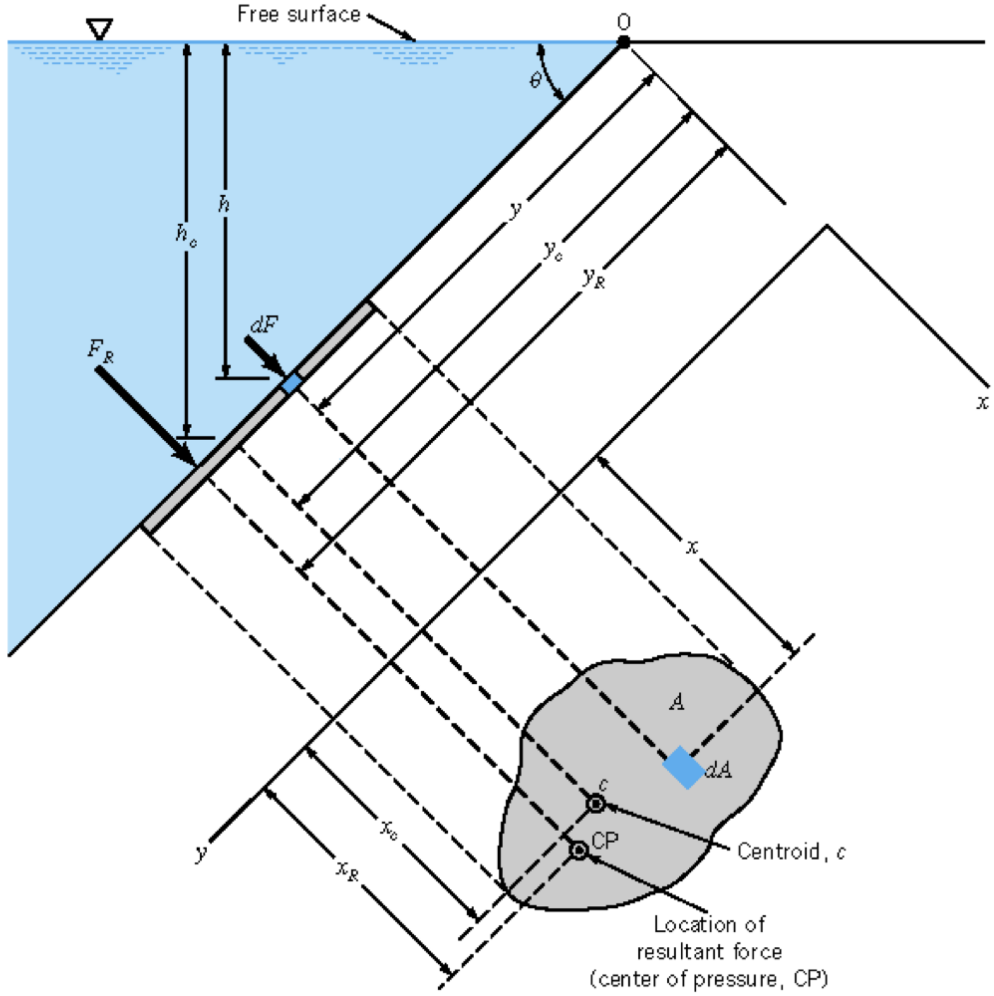


FIG. 2

Note that,

- The liquid surface is horizontal, as shown in the figure.
- The inclined object makes an angel θ with the free surface. The plane where the object lies define the y coordinate. The x coordinate is in the direction perpendicular to the object surface.

To compute the resultant force acting on the object F_R...

- First, we consider a small differential area dA of the object.
- At any given depth h, the force acting on dA is

  .

  Note that, this force is acting perpendicular to the surface in the x direction.
- Then, we can determine the magnitude of the resultant force by summing the forces on all these differential area over the entire surface of the object:

  .

  Note that we have used $h = y \sin\theta$.
- For constant ρ, g and θ, we can further write
- Next, we invoke the relation $\int_A y dA = y_c A$. The integral $\int_A y dA$ is the first moment of the area with respect to x; see notes for further discussion (optional). The derivation of this relation is not our focus. Rather our focus here is the quantity y_c, which is the distance of the centroid of the object measured from the origin O.
- To sum up, we can write:

  .

  or equivalent

  .

  which uses the relation $h_c = y_c \sin\theta$.
- Now we know the magnitude of the resultant force F_R acting on the a submerged object.

2 Computing hydrostatic force location, y_R

- There is still one question left to be answered: where on the object does this resultant force act on? Initially, we may think that the resultant force should act through the centroid of the object. However, this may not be true in general.
- Let's use the parameter y_R to denote the distance from the origin O to the location where the resultant force acts on the object. Our goal here is to find y_R.
- To this end, let's take moment about O. Remember that moment is defined as force times distance. This enables us to write

  .

  Physically, the LHS of the equation means that the resultant force acting on the object F_R (which is known by now) is acting on y_R (which we are going to determine). The RHS means that we are taking moments of the distributed force on the differential area of the object.
- Using our previous results, we can manipulate the equation as:

  .

  .
- The integral $\int_A y^2 dA$ is known as the second moment of area and let's call it I_0. The integral form of I_0 is not useful to our goal of finding y_R. However, I_0 can be written as $I_0 = I_c + Ay_c^2$, where we have invoked the parallel axis theorem. We will not get into the details of the theorem because it is purely a geometric exercise; see notes for further discussion (optional).
- To sum up, we have

  For a submerged object, usually y_c and A are known. I_c is a function of the shape of the object and is also known; see page 11 of the notes. Thus, this enables you to determine y_R.

FLUIDS IN MOTION (BOOK CHAPTER 3, 5, 8)

Fluid motions manifest themselves in many different ways. Some can be described very easily, while others require a thorough understanding of physical laws. In engineering applications, it is important to describe the fluid motions as simply as can be justified. It is the engineer's responsibility to know which simplifying assumptions (e.g., one-dimensional, steady-state, inviscid, incompressible, etc) can be made.

A. Classification of Fluid Flows

1) Uniform flow; steady flow

If we look at a fluid flowing under normal circumstances - a river for example - the conditions (e.g. velocity, pressure) at one point will vary from those at another point, then we have non-uniform flow. If the conditions at one point vary as time passes, then we have unsteady flow.

Under some circumstances the flow will not be as changeable as this. The following terms describe the states which are used to classify fluid flow:

Uniform flow: If the flow velocity is the same magnitude and direction at every point in the flow it is said to be uniform. That is, the flow conditions DO NOT change with **position**.

Non-uniform: If at a given instant, the velocity is not the same at every point the flow is non-uniform.

Steady: A steady flow is one in which the conditions (velocity, pressure and cross-section) may differ from point to point but DO NOT change with **time**.

Unsteady: If at any point in the fluid, the conditions change with time, the flow is described as unsteady.

Combining the above we can classify any flow in to one of four types:

- Steady uniform flow. Conditions do not change with position in the stream or with time. An example is the flow of water in a pipe of constant diameter at constant velocity.

- Steady non-uniform flow. Conditions change from point to point in the stream but do not change with time. An example is flow in a tapering pipe with constant velocity at the inlet - velocity will change as you move along the length of the pipe toward the exit.

- Unsteady uniform flow. At a given instant in time the conditions at every point are the same, but will change with time. An example is a pipe of constant diameter connected to a pump pumping at a constant rate which is then switched off.

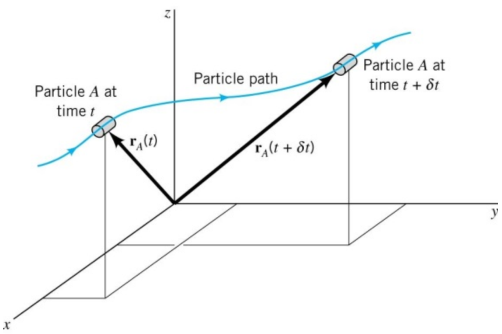
- Unsteady non-uniform flow. Every condition of the flow may change from point to point and with time at every point. An example is surface waves in an open channel.

You may imagine that one class is more complex than another – *steady uniform* flow is by far the most simple of the four.

2) One-, two-, and three-dimensional flows

A fluid flow is in general a three-dimensional, spatial and time dependent phenomenon:-

where $\vec{r} = (x, y, z)$ is the position vector, $(\vec{\boldsymbol{i}}, \vec{\boldsymbol{j}}, \vec{\boldsymbol{k}})$ are the unit vectors in the Cartesian coordinates, and (u, v, w) are the velocity components in these directions. As defined above, the flow will be uniform if the velocity components are independent of spatial position (x, y, z), and will be steady if the velocity components are independent of time *t*.



Accordingly, a fluid flow is called three-dimensional if all three velocity components are equally important. Intrinsically, a three-dimensional flow problem will have the most complex characters and is the most difficult to solve.

Fortunately, in many engineering applications, the flow can be considered as *two-dimensional*. In such a situation, one of the velocity components (say, *w*) is either identically zero or much smaller than the other two components, and the flow conditions vary essentially only in two directions (say, *x* and *y*). Hence, the velocity is reduced to $\boldsymbol{V} = u\vec{\boldsymbol{i}} + v\vec{\boldsymbol{j}}$ where (u, v) are functions of (x, y) (and possibly *t*). This reduction in the velocity component and spatial dimension will greatly simplify the analysis. Examples of two-dimensional flow typically involve flow past a <u>long</u> structure (with the axis of structure being perpendicular to the flow):

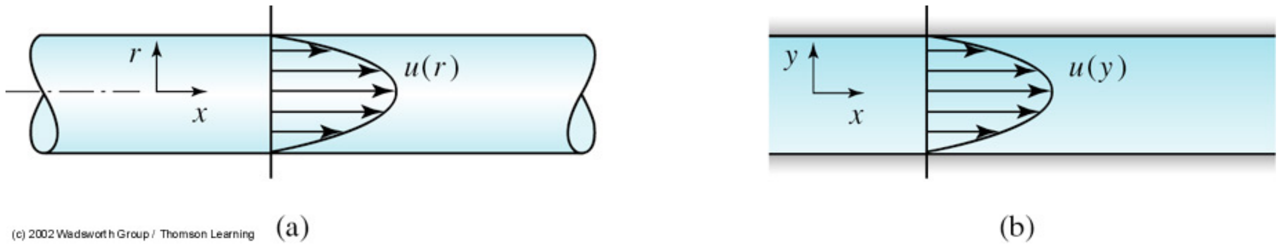
3) Viscous and inviscid flows

An **inviscid flow** is one in which viscous effects do not significantly influence the flow and are thus neglected. In a **viscous flow** the effects of viscosity are important and cannot be ignored.

To model an inviscid flow *analytically*, we can simply let the viscosity be zero; this will obviously make all viscous effects zero. It is more difficult to create an inviscid flow *experimentally*, because all fluids of interest (such as water and air) have viscosity. The question then becomes: are there flows of interest in which the viscous effects are negligibly small? The answer is "yes, if the shear stresses in the flow are small and act over such small areas that they do not significantly affect the flow field." The statement is very general, of course, and it will take considerable analysis to justify the inviscid flow assumption.

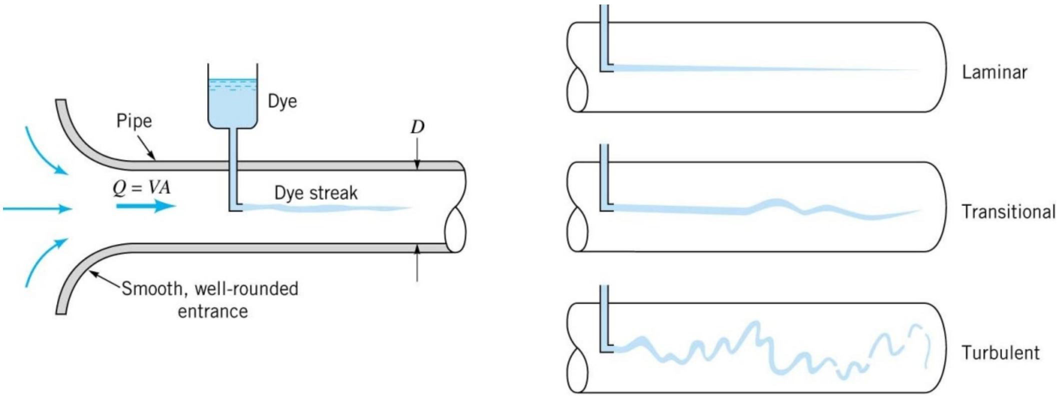
Based on experience, it has been found that the primary class of flows, which can be modeled as inviscid flows, is **external flows**, that is, flows of an unbounded fluid which exist exterior to a body. Inviscid flows are of primary importance in flows around *streamlined* bodies, such as flow around an airfoil (see the sketch below) or a hydrofoil. Any viscous effects that may exist are confined to a thin layer, called a **boundary layer**, which is attached to the boundary, such as that shown in the figure; the velocity in a boundary layer is always zero at a fixed wall, a result of viscosity. For many flow situations, boundary layers are so thin that they can simply be ignored when studying the gross features of a flow around a streamlined body. For example, the inviscid flow solution provides an excellent prediction to the flow around the airfoil, except possibly near the trailing edge where **flow separation** may occur. However the boundary layers must be accounted for when the skin friction force on the body is to be calculated.

Viscous flows include the broad class of **internal flows**, such as flows in pipes, hydraulic machines, and conduits and in open channels. In such flows viscous effects cause substantial "losses" and account for the huge amounts of energy that must be used to transport oil and gas in pipelines. The no-slip condition resulting in zero velocity at the wall, and the resulting shear stresses, lead directly to these losses.



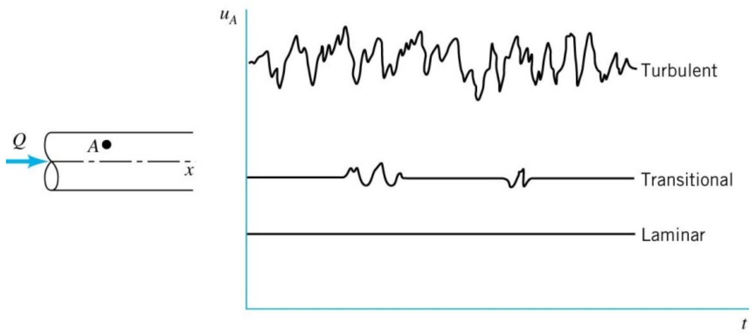
Viscous internal flow: (a) in a pipe; (b) between two parallel plates.

4) Laminar and turbulent flows



(a) (b)

In the experiment shown above, a dye is injected into the middle of pipe flow of water. The dye streaks will vary, as shown in (b), depending on the flow rate in the pipe. The top situation is called **laminar flow**, and the lower is **turbulent flow**, occurring when the flow is sufficiently slow and fast, respectively. In laminar flow the motion of the fluid particles is very orderly with all particles moving in straight lines parallel to the pipe wall. There is essentially no mixing of neighboring fluid particles. In sharp contrast, mixing is very significant in turbulent flow, in which fluid particles move haphazardly in all directions. It is therefore impossible to trace motion of individual particles in turbulent flow. The flow may be characterized by an unsteady fluctuating (i.e., random and 3-D) velocity components superimposed on a temporal steady mean (i.e., along the pipe) velocity.



Time dependence of fluid velocity at a point.

Whether the flow is laminar or not depends on the Reynolds number,

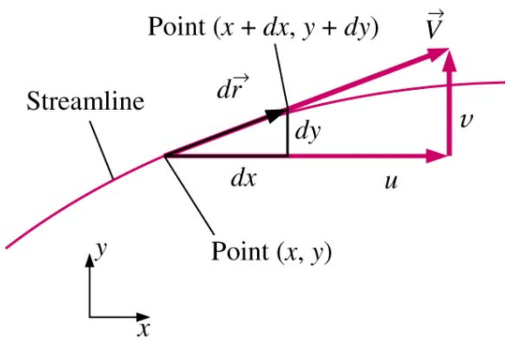
and it has been demonstrated experimentally that

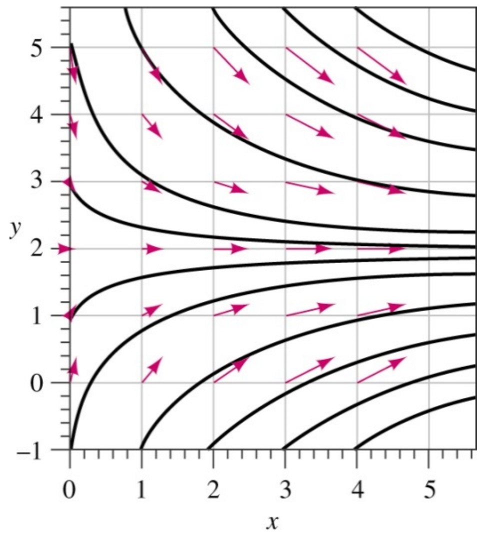
B. Flow Visualization

There are four different types of flow lines that may help to describe a flow field.

1) Streamline

A **streamline** is a line that is everywhere tangent to the velocity vector at a given instant of time. A streamline is hence an instantaneous pattern, i.e. a streamline changes with time in general.





Equation for a streamline

Example: Consider that velocity components are given by u = sin(3yt) and v = k, where t is time and k is a constant. We could substitute these components into the above equation and integrate both sides of the equation. The resulting equation that relates x and y is the equation for the streamline.

Streamlines are very useful to help visualize the flow pattern. Another example of the streamlines around a cross-section of an airfoil has been shown earlier on page 3.

When fluid is flowing past a solid boundary, e.g., the surface of an aerofoil or the wall of a pipe, fluid obviously does not flow into or out of the surface. So very close to a boundary wall the flow direction must be parallel to the boundary. In fact, the boundary wall itself is also a streamline by definition.

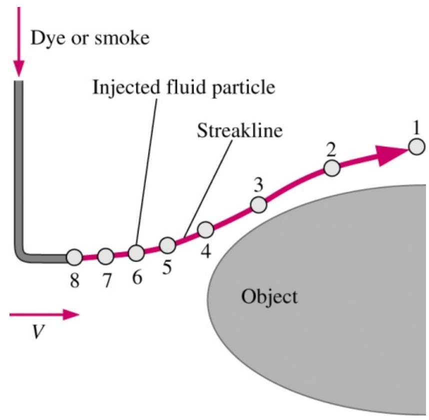
It is also important to recognize that the position of streamlines can change with time - this is the case in unsteady flow. In steady flow, the streamlines do not change.

Some further remarks about streamlines

- Because the fluid is moving in the same direction as the streamlines, fluid cannot cross a streamline.
- Streamlines cannot cross each other. If they were to cross, this would indicate two different velocities at the same point. This is not physically possible.
- The above point implies that any particles of fluid starting on one streamline will stay on that same streamline throughout the fluid.

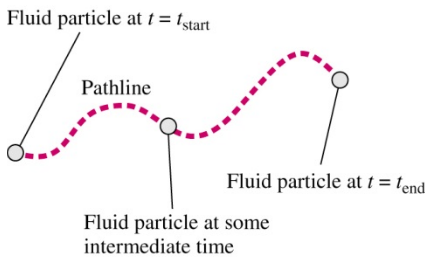
2) Streakline

A **streakline** is an instantaneous line whose points are occupied by particles which have earlier passed through a prescribed point in space. A streakline is hence an integrated pattern. A streakline can be formed by injecting dye continuously into the fluid at a fixed point in space. As time marches on, the streakline gets longer and longer, and represents an integrated history of the dye streak.



3) Pathline

A **pathline** is the actual path traversed by a given (marked) fluid particle. A pathline is hence also an integrated pattern. A pathline represents an integrated history of where a fluid particle has been.



For steady flow, streamlines, streaklines, and pathlines are all identical. However, for unsteady flow, these three flow patterns can be quite different. In a steady flow, all particles passing a given point will continue to trace out the same path since nothing changes with time; hence the pathlines and streaklines coincide. In addition, the velocity vector of a particle at a given point will be tangent to the line that the particle is moving along; thus the line is also a streamline.

C. <u>Elementary Equations of Motion</u>

In analyzing fluid motion, we might take one of two approaches: (1) seeking to describe the detailed flow pattern at every point (*x,y,z*) in the field, or (2) working with a finite region, making a balance of *flow in* versus *flow out*, and determining gross flow effects such as the force, or torque on a body, or the total energy exchange. The second approach is the "**control-volume**" method and is the subject of this section. The first approach is the "**differential**" approach and will be covered in the later part of this course.

We shall derive the three basic control-volume relations in fluid mechanics:

- the principle of conservation of mass, from which the continuity equation is developed;
- the principle of conservation of energy, from which the energy equation is derived;
- the principle of conservation of linear momentum, from which equations evaluating dynamic forces exerted by flowing fluids may be established.

1) Control volume

- A control volume is a finite region, chosen carefully by the analyst for a particular problem, with open boundaries through which mass, momentum, and energy are allowed to cross. The analyst makes a budget, or balance, between the incoming and outgoing fluid and the resultant changes within the control volume. Therefore one can calculate the <u>gross properties</u> (net force, total power output, total heat transfer, etc.) with this method.
- With this method, however, we do not care about the details inside the control volume (In other words we can treat the control volume as a "black box.")
- For the sake of the present analysis, let us consider a control volume that can be a tank, reservoir or a compartment inside a system, and consists of some definite *one-dimensional* inlets and outlets, like the one shown below:

  Let us denote for each of the inlets and outlets:-

  V = velocity of fluid in a stream
  A = sectional area of a stream
  p = pressure of the fluid in a stream
  ρ = density of the fluid

  Then, the **volume flow rate**, or **discharge** (volume of flow crossing a section per unit time) is given by

  Similarly, the **mass flow rate** (mass of flow crossing a section per unit time) is given by

  Then, the **momentum flux**, defined as the momentum of flow crossing a section per unit time, is given by

- For simplicity, we shall from here on consider **steady** and **incompressible** flows only.

2) Continuity equation

By steadiness, the total mass of fluid contained in the control volume must be invariant with time. Therefore there must be an exact balance between the total rate of flow into the control volume and that out of the control volume:

Total Mass Outflow = Total Mass Inflow

which translates into the following mathematical relation

where *M* is the number of inlets, and *N* is the number of outlets. If the density of fluid is constant, conservation of mass also implies conservation of volume. Hence for a control volume with only one-dimensional inlets and outlets,

For example, in a pipe of varying cross sectional area, the continuity equation requires that, if the density is constant, between any two sections 1 and 2 along the pipe

Another example involving two inlets and one outlet is shown below.

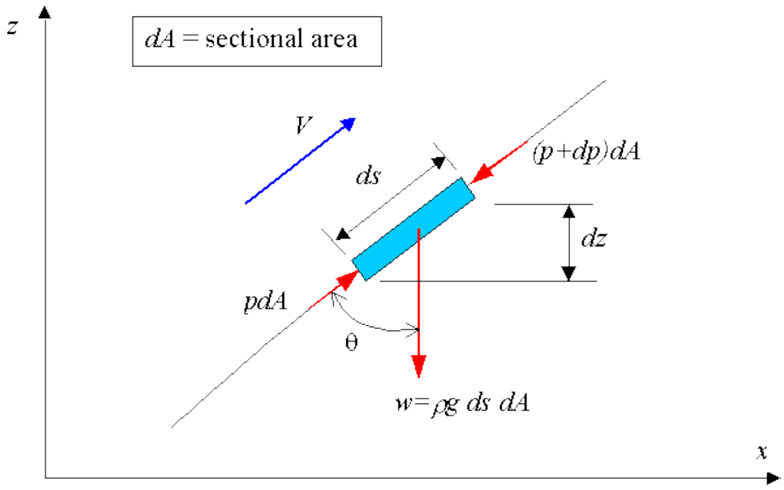
3) Bernoulli and energy equations

Let us first derive the *Bernoulli* equation, which is one of the most well-known equations of motion in fluid mechanics, and yet is often misused. It is thus important to understand its limitations, and the assumptions made in the derivation.

The assumptions can be summarized as follows:

- Inviscid flow (ideal fluid, frictionless)
- Steady flow (unsteady Bernoulli equation will not be discussed in this course)
- Along a streamline
- Constant density (incompressible flow)
- No shaft work or heat transfer

The Bernoulli equation is based on the application of Newton's law of motion to a fluid element on a streamline.



Let us consider the motion of a fluid element of length ds and cross-sectional area dA moving at a local speed V, and x is a horizontal axis and z is pointing vertically upward. The forces acting on the element are the pressure forces pdA and $(p+dp)dA$, and the weight w as shown. Summing forces in the direction of motion, the s-direction, there results

$$pdA-(p+dp)dA-\rho g\ ds\ dA\cos\theta=\rho\ ds\ dA\ a_s$$

where a_s is the acceleration of the element in the s-direction. Since the flow is steady, only convective acceleration exists

$$a_s=V\frac{dV}{ds}$$

Also, it is easy to see that $\cos\theta=dz/ds$. On substituting and dividing the equation by $\rho g dA$, we can obtain **Euler's equation:**

$$\frac{dp}{\rho g}+dz+\frac{V}{g}dV=0$$

Note that Euler's equation is valid also for compressible flow.

Now if we further assume that the flow is incompressible so that the density is constant, we may integrate Euler's equation to get

$$\boxed{\frac{p}{\rho g} + z + \frac{V^2}{2g} = \text{constant}}$$

This is the Bernoulli equation, consisting of three **energy heads**

$\frac{p}{\rho g}$	Pressure head, which is the work done to move fluid against pressure
z	Elevation head, representing the potential energy; z can be measured above any reference datum
$\frac{V^2}{2g}$	Velocity head, representing the kinetic energy

- A **head** corresponds **to energy per unit weight of flow** and has dimensions of **length**.
- **Piezometric head = pressure head + elevation head**, which is the level registered by a piezometer connected to that point in a pipeline.
- **Total head = piezometric head + velocity head.**

It follows that for ideal steady flow the total energy head is constant along a streamline, but the constant may differ in different streamlines. (For the particular case of irrotational flow, the Bernoulli constant is universal throughout the entire flow field.)

Applying the Bernoulli equation to any two points on the same streamline, we have

There is similarity in form between the Bernoulli equation and the energy equation that can be derived directly from the first law of thermodynamics. Without getting into the derivation, the energy equation for a control volume with only one inlet (section 1) and one outlet (section 2) can be written as

where $\dot{W}_s$ is the **shaft work**, or the rate of work transmitted by rotation shafts (such as that of a pump or turbine; positive if output to a turbine, negative if input by a pump) that are cut by the control surface, and h_L, called the **head loss**, is the sum of energy losses required to overcome viscous forces in the fluid (dissipated in the form of thermal energy) and the heat transfer rate. In the absence of these two terms, the energy equation is identical to the Bernoulli equation. We must remember however that the Bernoulli equation is a momentum equation applicable to a streamline and the energy equation above is applied between two sections of a flow. The energy equation is more general than the Bernoulli equation, because it allows for (1) friction, (2) heat transfer, (3) shaft work, and (4) viscous work (another frictional effect).

4) Momentum equation

On applying Newton's second law of motion to the control volume shown on page 8, we get

Note that this equation

- follows from the principle of conservation of linear momentum: resultant force on the control volume is balanced by the net change of momentum flux (i.e., $\dot{m}\vec{V}$) on getting out through the control surface.
- is a vector equation. **Components** of the forces and the velocities need to be considered.
- can be used to calculate the magnitude and direction of the impact force exerted on the control volume by its solid boundary.

Further consider a steady-flow situation in which there is only one entrance (section 1) and one exit (section 2) across which uniform profiles can be assumed (see the figure on page 9). By continuity

The momentum equation now reduces to

or in terms of their components in (x, y, z) coordinates

where $(V_x)_1$ is the x-component of the velocity at section 1, and so on.

On applying the momentum equation, one needs to pay attention to the following two aspects.

Forces

$\sum \vec{F}$ represents **all forces acting on the control volume**, including

- Surface forces resulting from the surrounding acting on the control volume:
  - Impact force, which is usually the unknown to be found, on the control surface in contact with a solid boundary
  - Pressure force on the control surface which cuts a flow inlet or exit. Remember that the pressure force is always a compressive force.
- Body force that results from gravity.

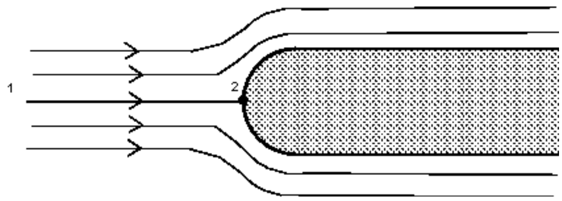
Sign of the vector variables

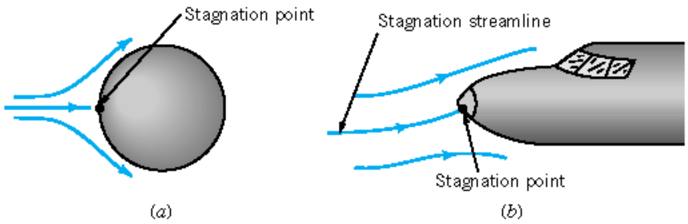
When plugging into the equations, one should be careful about the sign of the force and velocity components. These quantities should carry a positive (negative) sign when they are in the same (opposite) sense as that of the corresponding coordinate.

D. Applications of the Bernoulli and Momentum Equations

1) Pitot tube

If a stream of uniform velocity flows into a blunt body, the streamlines take a pattern similar to this:





Streamlines around blunt bodies

Note how some move to the left and some to the right. But one, in the center, goes to the tip of the blunt body and stops. It stops because at this point the velocity is zero - the fluid does not move at this one point. This point is known as the **stagnation point**.

From the Bernoulli equation we can calculate the pressure at this point. Apply Bernoulli equation along the central streamline from a point upstream where the velocity is V_1 and the pressure p_1 to the stagnation point of the blunt body where the velocity is zero, $V_2 = 0$. Also $z_1 = z_2$.

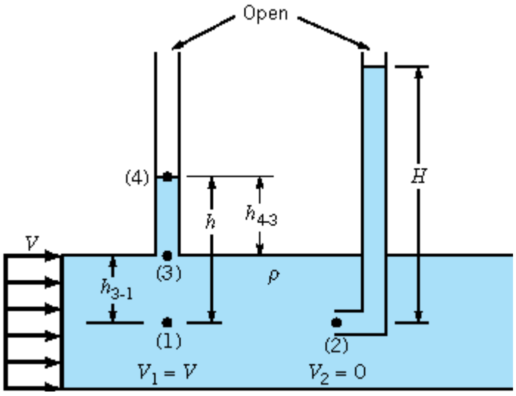
This increase in pressure, which brings the fluid to rest, is called the *dynamic pressure*.

Dynamic pressure =
or converting this to head (using $h = p / \rho g$)
Dynamic head =

The total pressure is known as the *stagnation pressure (or total pressure)*

Stagnation pressure =
or in terms of head,
Stagnation head =

The blunt body stopping the fluid does not have to be a solid. It could be a static column of fluid. Two piezometers, one as normal and one as a Pitot tube within the pipe can be used in an arrangement shown below to measure velocity of flow.



A Piezometer and a Pitot tube.

Using the above theory, we have the equation for p_2,

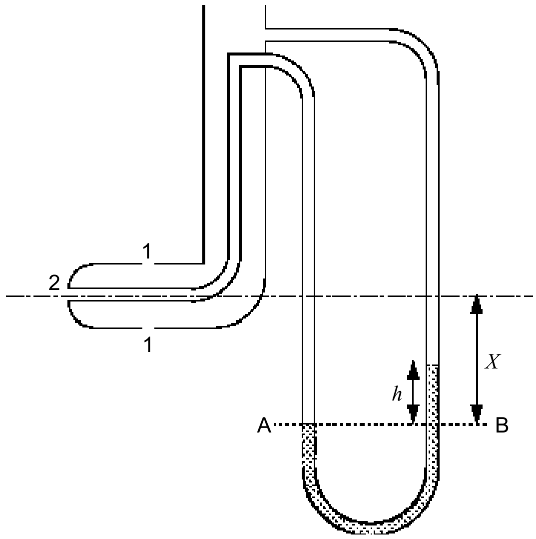
which is an expression for velocity obtained from two pressure measurements and the application of the Bernoulli equation. This equation is for ideal flow only. To account for real fluid effects, the equation can be modified into $V = C_v\sqrt{2g\left(H-h\right)}$, where C_v is the coefficient of velocity to be determined empirically.

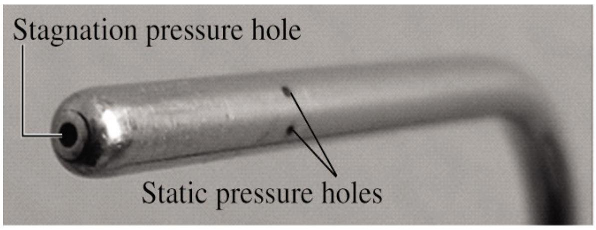
A Pitot tube used to measure velocity of flow in a channel.

A Pitot tube underneath the wing of an aircraft.

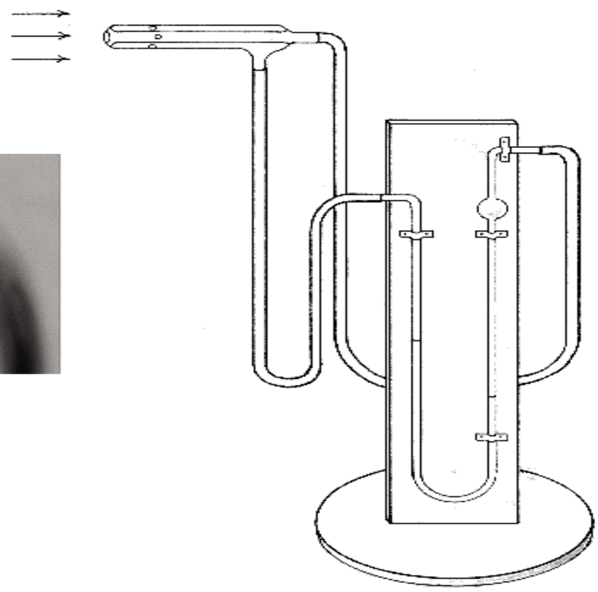
2) Pitot static tube

The necessity of one piezometer and one Pitot tube and thus two readings make this arrangement a little awkward. Connecting the two tubes to a manometer would simplify things but there are still two tubes. The **Pitot static tube** combines the tubes, and they can then be easily connected to a differential manometer. A Pitot static tube is shown here. The holes on the side of the tube connect to one side of a manometer and register the *static head*, (h_1), while the central hole is connected to the other side of the manometer to register, as before, the *stagnation head* (h_2). The difference of the two heads, being the *dynamic head*, is now measured directly by the differential manometer.





Close-up of a Pitot static tube.



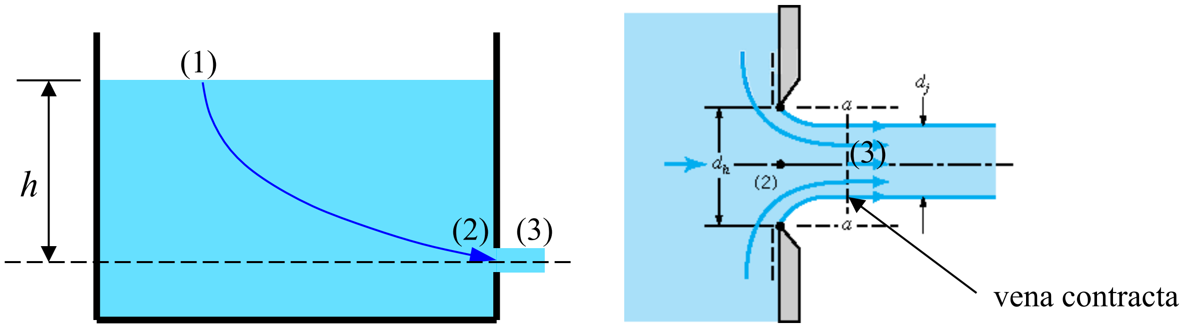
Consider the pressures on the level of the centre line of the Pitot static tube and using the theory of the manometer,

We also know that $p_2 = p_1 + \rho V^2 / 2$. Hence

The Pitot/Pitot-static tubes give velocities at points in the flow. It does not give the overall discharge of the stream, which is often what is wanted. It also has the drawback that it is liable to block easily, particularly if there is significant debris in the flow.

3) Orifice and vena contracta

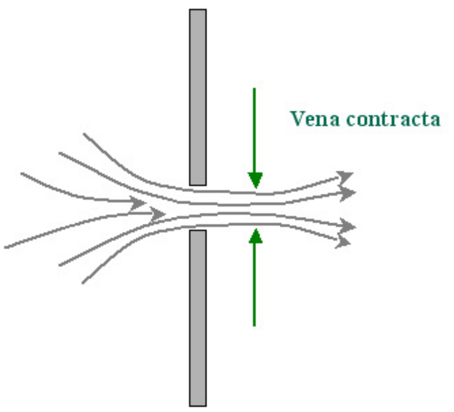
We are to consider the flow from a tank through a hole on a side wall. The general arrangement and a close-up of the hole and streamlines are shown in the figure below.

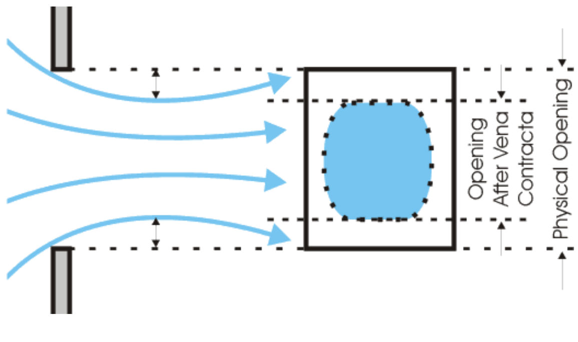


Tank and streamlines of flow out of a sharp-edged orifice

The hole is sharp-edged so as to minimize frictional losses by minimizing the contact between the hole and the liquid issuing from the hole.

Looking at the streamlines you can see how they contract after the orifice to a minimum cross section where they all become parallel, at this point, the velocity and pressure are uniform across the jet. This convergence is called the **vena contracta** (from the Latin 'contracted vein'). It is necessary to know the amount of contraction to allow us to calculate the flow.





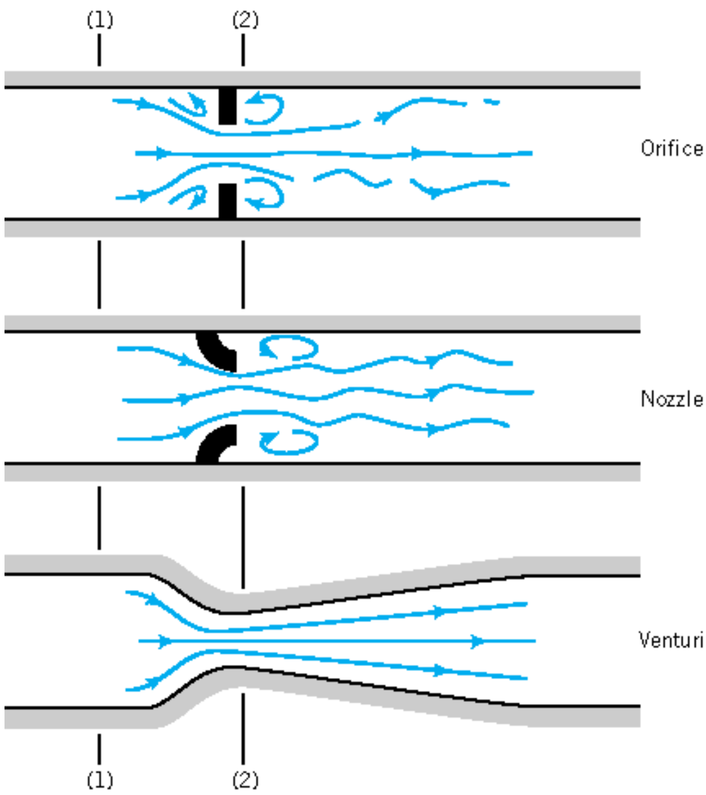
We can predict the velocity at the orifice using the Bernoulli equation. Apply it along the streamline joining point 1 on the surface to point 3 at the centre of the vena contracta.

At the surface velocity is negligible ($V_1 = 0$) and the pressure atmospheric ($p_1 = 0$). Outside the orifice the jet is open to the air so again the pressure is atmospheric ($p_3 = 0$). If we take the datum line through the orifice then $z_1 = h$ and $z_3 = 0$, leaving

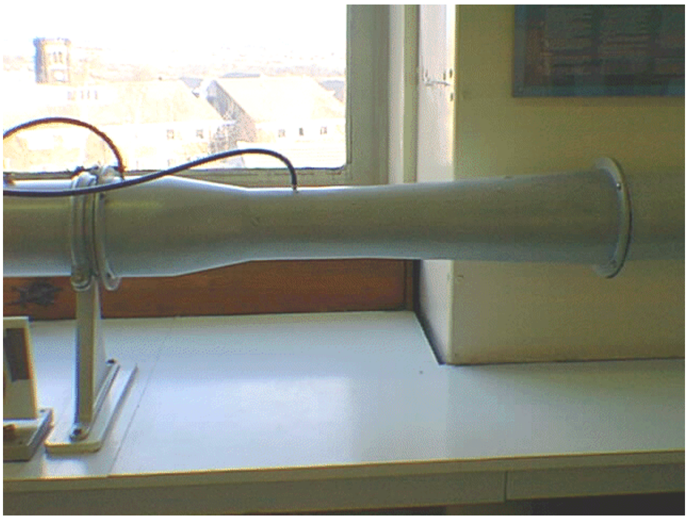
4) Venturi, nozzle and orifice meters

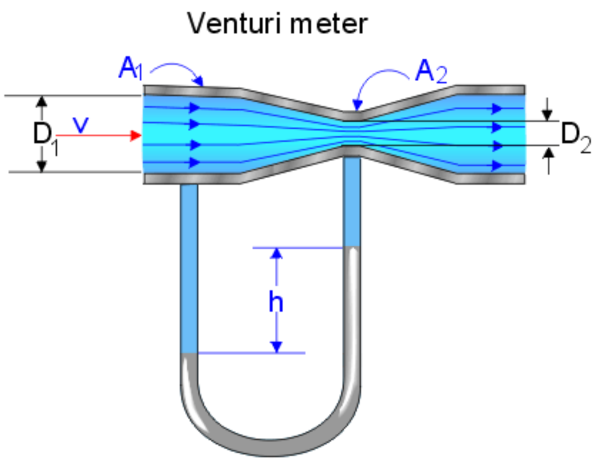
The Venturi-, nozzle- and orifice-meters are three similar types of devices for measuring discharge in a pipe. The Venturi meter consists of a rapidly converging section, which increases the velocity of flow and hence reduces the pressure. It then returns to the original dimensions of the pipe by a gently diverging 'diffuser' section. By measuring the pressure differences the discharge can be calculated. This is a particularly accurate method of flow measurement as energy losses are very small.

The nozzle meter or flow nozzle is essentially a Venturi meter with the convergent part replaced by a nozzle installed inside the pipe and the divergent part omitted. The orifice meter is a still simpler and cheaper arrangement by which a sharp-edged orifice is fitted concentrically in the pipe.



Schematic arrangements for three types of devices measuring flow-rate in a pipe





A Venturi meter in laboratory.

The working formulae are similar for the three devices. Let us for illustration show the one for the Venturi meter. Applying the Bernoulli equation along the streamline from point 1 to point 2 in the narrow *throat* of the Venturi meter, we have

$$\frac{p_1}{\rho g} + z_1 + \frac{V_1^2}{2g} = \frac{p_2}{\rho g} + z_2 + \frac{V_2^2}{2g}$$

By using the continuity equation we can eliminate the velocity V_2, $Q = A_1V_1 = A_2V_2$ or $V_2 = A_1V_1 / A_2$.

Substituting this into and rearranging the Bernoulli equation we get

To get the theoretical discharge this is multiplied by the area. To get the actual discharge taking in to account the losses due to friction, we include a coefficient of discharge

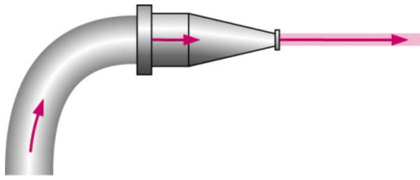
Suppose a differential manometer is connected between (1) and (2). Then the terms inside the square brackets can be related to the manometer reading h as given by

Thus the discharge can be expressed in terms of the manometer reading:

It should be noted that in deriving a formula for a discharge measuring device (Venturi, nozzle, orifice meters, etc), assumptions are taken to simplify the situations so that the Bernoulli equation can be applied. For example, there is no energy loss and the flow is steady. In this way, exact analytical solutions can be obtained, but as the assumptions are not exactly true, these solutions fail to account for the real situations. Empirical coefficients such as C_v, C_d are therefore introduced to allow for these errors. The final formula will be an analytical solution modified by an empirical coefficient. On the other hand, the value of the empirical coefficient can also reflect the justification of using the ideal approach. C_d for orifice meter is far below unity (0.6-0.65), while C_d for nozzles and venturi meters are close to one (approximately 0.98). It shows that energy loss is rather substantial in an orifice meter, as is expected from its abrupt configuration.

5) Force on a pipe nozzle

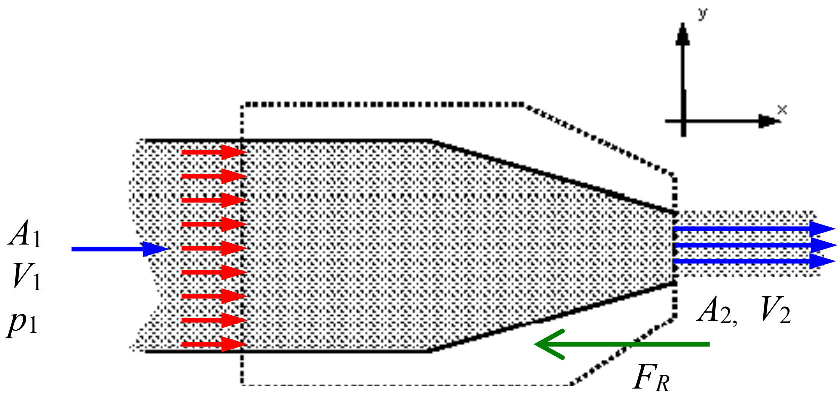
Let us from here on consider several applications of the momentum equations. A simple application is to find the force on the nozzle at the outlet of a pipe. Because the fluid is contracted at the nozzle forces are induced in the nozzle. Anything holding the nozzle (e.g. a fireman) must be strong enough to withstand these forces.



Steps in analysis:

1. Draw a control volume
2. Decide on a coordinate-axis system
3. Calculate the rate of change of momentum across the control volume
4. Calculate the pressure force F_p
5. Calculate the body force F_B
6. Calculate the resultant reaction force F_R

1 & 2. <u>Control volume and co-ordinate axis are shown in the figure below</u>.



Notice how this is a one-dimensional system which greatly simplifies matters.

3. <u>Calculate the change of momentum flux</u> (RHS of the momentum equation)

By continuity, $Q = A_1V_1 = A_2V_2$, so

4. <u>Calculate the pressure force</u> (red arrows)

$$F_p = \text{ pressure force at 1 } - \text{ pressure force at 2} = p_1A_1 - p_2A_2$$

We use the Bernoulli equation to calculate the pressure

$$\frac{p_1}{\rho g} + z_1 + \frac{V_1^2}{2g} = \frac{p_2}{\rho g} + z_2 + \frac{V_2^2}{2g}$$

Since the nozzle is horizontal, $z_1 = z_2$, and the pressure outside is atmospheric, $p_2 = 0$, and with continuity the Bernoulli equation gives

5. Calculate the **body** force

The only body force is the weight due to gravity in the y-direction - but we need not consider this as the only forces we are considering are in the x-direction.

6. Calculate the **reaction** force that the nozzle acts on the fluid (green arrow)

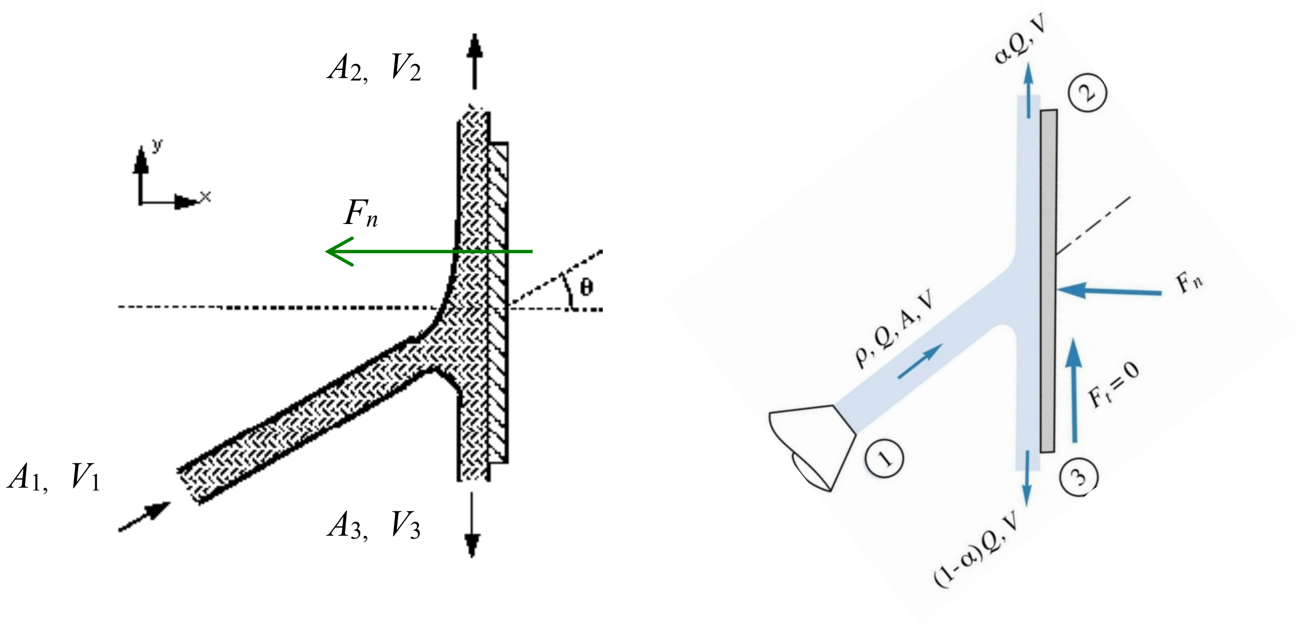
Since the indicated direction of the reaction force is opposite to x-axis, a negative sign is included

So the fireman must be able to resist the force of F_R.

6) Force due to a two-dimensional jet hitting an inclined plane

Consider a two-dimensional (i.e., very wide in the spanwise direction) jet hitting a flat plate at an angle θ. For simplicity gravity and friction are neglected from this analysis.

We want to find the reaction force normal to the plate so we choose the axis system such that it is normal to the plane.



A two-dimensional jet hitting an inclined plate.

We do not know the velocities of flow in each direction. To find these we can apply the Bernoulli equation

The height differences are negligible i.e., $z_1 = z_2 = z_3$, and the pressures are all atmospheric = 0. So

By continuity

Using this we can calculate the forces in the same way as before.

1. Calculate the total force in the *x*-direction.

Remember that the co-ordinate system is normal to the plate.

but $V_{2x} = V_{3x} = 0$ as the jets are parallel to the plate with no component in the x-direction, and $V_{1x} = V\cos\theta$, so

2. Calculate the pressure force

All zero as the pressure is everywhere atmospheric.

3.Calculate the body force

As the control volume is small, hence the weight of fluid is small, we can ignore the body forces.

4. Calculate the resultant reaction force

which is the force exerted **on the fluid by the plate**.

We can further find out how much discharge goes along in each direction on the plate. Along the plate, in the *y*-direction, the total force must be zero, $\sum F_y = 0$, since friction is ignored.

Also in the y-direction: $V_{1y} = V\sin\theta$, $V_{2y} = V$, $V_{3y} = -V$, so

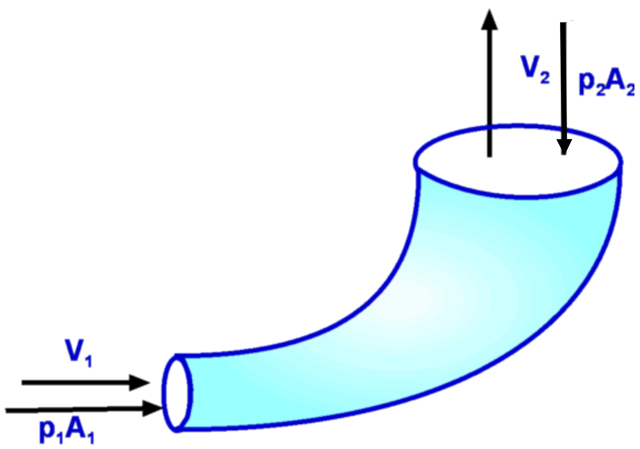
Setting this to zero, we get

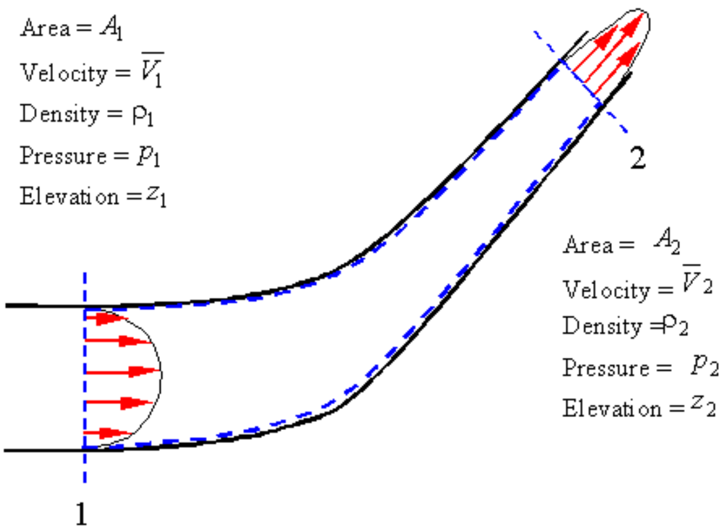
and as found earlier we have $A_1 = A_2 + A_3$, so on solving

by which we readily obtain that

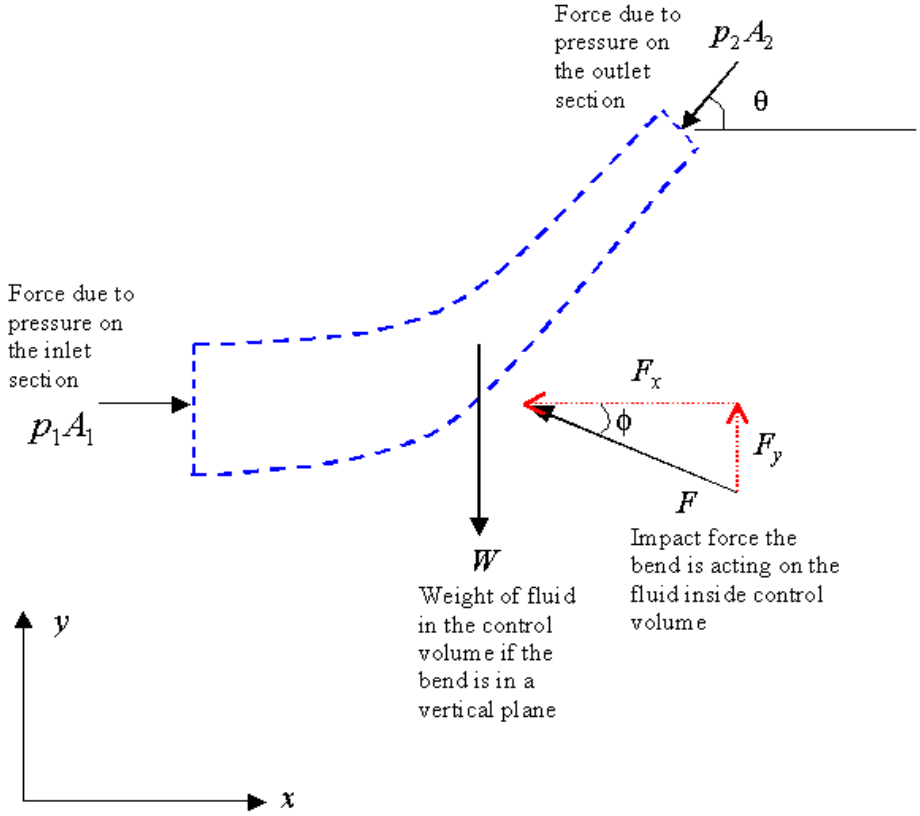
So we know how the discharge is divided between the two jets leaving the plate.

7) Flow past a pipe bend





Consider the pipe bend shown above. We may first draw a free body diagram for the control volume with the forces:



Paying due regard to the positive x and y directions, we may write the summation of forces in these two directions:

$$\sum F_x = p_1 A_1 - p_2 A_2 \cos\theta - F_x$$
$$\sum F_y = F_y - p_2 A_2 \sin\theta - W$$

Relating these components to the net change of momentum flux through the inlet and exit surfaces

x-Direction

$$p_1 A_1 - p_2 A_2 \cos\theta - F_x = \rho Q\left(\bar{V}_2 \cos\theta - \bar{V}_1\right)$$

y-Direction

$$F_y - p_2 A_2 \sin\theta - W = \rho Q\left(\bar{V}_2 \sin\theta - 0\right)$$

From these two equations and using the continuity equation and the Bernoulli equation, we may calculate the two force components. The magnitude and direction of the resultant force from the bend on the fluid are

$$F = \sqrt{F_x^2 + F_y^2}$$
$$\phi = \tan^{-1}\left(F_y / F_x\right)$$

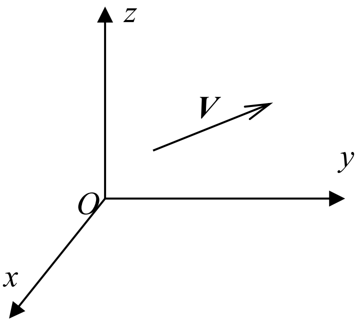
As a reaction, the impact force on the pipe bend is equal in magnitude, but opposite in direction to the one on the fluid.

(V) FLUID KINEMATICS & DIFFERENTIAL ANALYSIS OF FLUID FLOW (BOOK CHAPTER 4 & 6)

A. Description of Fluid Motion

- Lagrangian description: fluid particles are "tagged" or identified; rate of change of flow properties as observed by following a particle of fixed identity; variables are functions of the initial position of particles and time.
- Eulerian description: fluid properties and variables are field variables, which are functions of position in space (with respect to a fixed frame of reference) and time. The Eulerian description, which is comparable to the data recorded by a measuring device fixed in position, is more convenient to use in fluid mechanics.

- Rectangular (Cartesian) coordinates:



$$\boldsymbol{x} = (x, y, z) = (x_1, x_2, x_3) = x_i \quad (i = 1,2,3)$$

$$\boldsymbol{V} = (u, v, w) = (u_x, u_y, u_z) = (u_1, u_2, u_3) = u_i \quad (i = 1,2,3)$$

$$\nabla = \left(\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z}\right) = \left(\frac{\partial}{\partial x_1}, \frac{\partial}{\partial x_2}, \frac{\partial}{\partial x_3}\right) = \frac{\partial}{\partial x_i} \quad (i = 1,2,3) \quad \text{"del"}$$

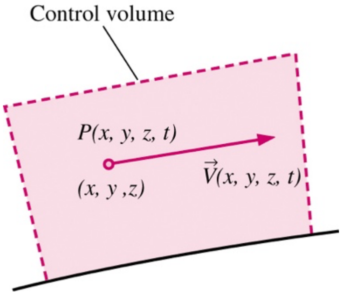
e.g., $$\nabla \cdot \boldsymbol{V} = \left(\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z}\right) \cdot (u, v, w) = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = \frac{\partial u_i}{\partial x_i}$$

(NB: $\nabla \cdot \boldsymbol{V} \neq \boldsymbol{V} \cdot \nabla$)

A scalar is a quantity with magnitude only. A vector is a quantity with both magnitude and direction, hence with a single free index. A second-order tensor is a quantity with magnitude and associated with two directions, hence with two free indices. According to the Einstein summation convention, when an index appears twice in a single term it implies summation of that term over all the values of the index. A repeated index is called a dummy index.

- Primitive variables: pressure $p(\boldsymbol{x}, t)$ - scalar (0th order tensor)
  velocity $\boldsymbol{V}(\boldsymbol{x}, t)$ - vector (1st order tensor)

  Deduced variable stress $\boldsymbol{\tau}(\boldsymbol{x}, t)$ - 2nd order tensor



In the Eulerian description, one defines field variables, such as the pressure field and the velocity field, at any location and instant in time.

B. Kinematics

- Total (a.k.a. material, substantial) derivative = local rate of change + convective (or advective) rate of change = the rate of change as observed following a particle of fixed identity. It is an operator that can be applied to any scalar or vector quantity.

$$\frac{d(\;)}{dt} = \frac{\partial(\;)}{\partial t} + (\boldsymbol{V}\cdot\nabla)(\;)$$

$$= \underbrace{\frac{\partial(\;)}{\partial t}}_{\text{local rate of change}} + \underbrace{u\frac{\partial(\;)}{\partial x} + v\frac{\partial(\;)}{\partial y} + w\frac{\partial(\;)}{\partial z}}_{\text{convective rate of change}}$$

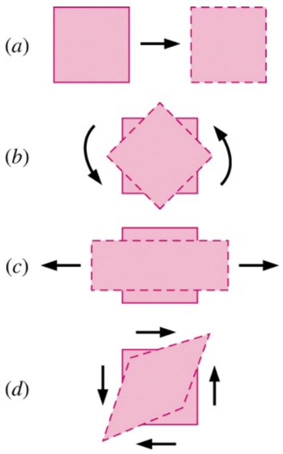
e.g., local acceleration = $\dfrac{\partial \boldsymbol{V}}{\partial t}$

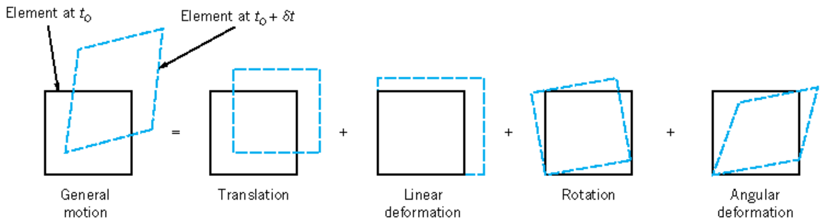
convective acceleration = $(\boldsymbol{V}\cdot\nabla)\boldsymbol{V} = u\dfrac{\partial \boldsymbol{V}}{\partial x} + v\dfrac{\partial \boldsymbol{V}}{\partial y} + w\dfrac{\partial \boldsymbol{V}}{\partial z}$

- The local rate of change, also called the unsteady term, vanishes identically for a steady flow. Therefore a flow is steady if and only if $\partial/\partial t \equiv 0$.
- The quantity $(\boldsymbol{V}\cdot\nabla)$ is a scalar convective operator that determines the time rate of change of any property (e.g., velocity, density, concentration, temperature) of a particle by reason of the fact that the particle moves from a place where the property has one value to another place where it has a different value.

- $$\text{General motion} = \begin{cases} \left.\begin{array}{c}\text{Translation (trivial)} \\ + \\ \text{Rotation}\end{array}\right\} & \text{(rigid body motion)} \\ + & \\ \text{Dilatation} & \text{(change in volume)} \\ + & \\ \text{Angular deformation} & \text{(change in shape)} \end{cases}$$

As illustrated by:-

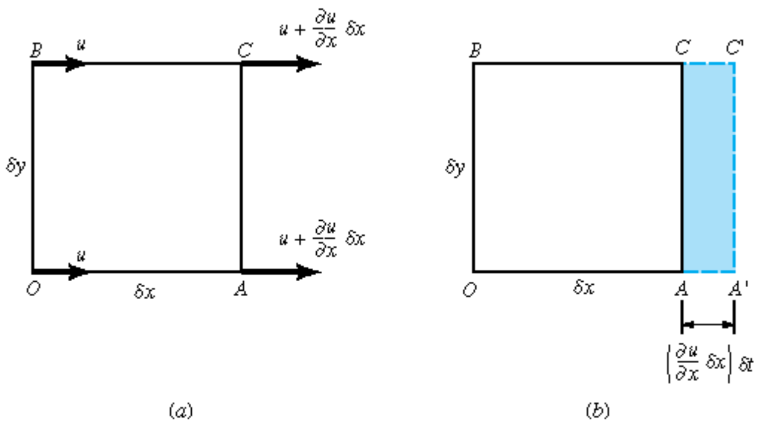




The various modes of deformation can be expressed in terms of the velocity gradients.

- Divergence of velocity is the volumetric strain/dilatation rate (rate of change of volume per unit volume)

where $\partial u/\partial x$, $\partial v/\partial y$ and $\partial w/\partial z$ are the components of the volumetric strain rate due to elongation of a fluid element in the x-, y-, and z-directions, respectively.
Consider a small element of dimensions $\delta x \times \delta y \times \delta z$:



Because of the velocity differential δu over a distance δx, the element is lengthened in the x-direction by $\delta u \cdot \delta t$ over a small period of time δt. The corresponding change in volume is therefore

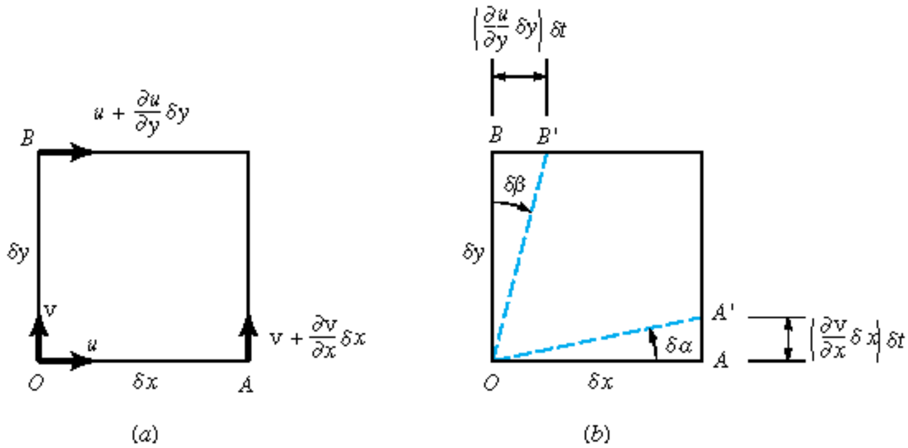
and the volume strain rate (change in volume per volume per time) is

Similarly, for the lengthening of the element in the y- and z-directions

$$\frac{\delta \forall_y}{\forall \cdot \delta t} = \frac{\partial v}{\partial y}, \qquad \frac{\delta \forall_z}{\forall \cdot \delta t} = \frac{\partial w}{\partial z} \quad \text{as } \delta y,\ \delta z,\ \delta t \to 0$$

The total volume strain rate is hence given by the divergence of the velocity.

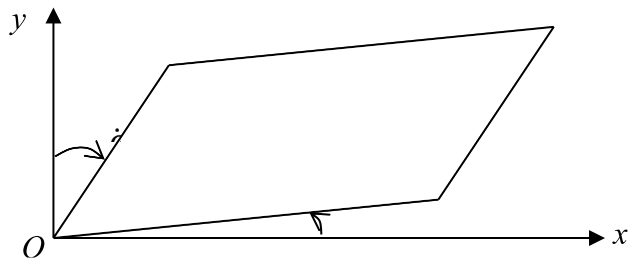
- Any shear deformation can be decomposed into rigid body rotation and angular deformation. Consider a small element undergoing shear deformation



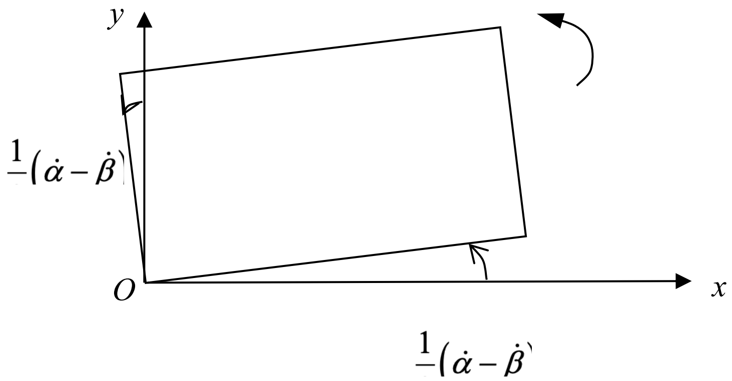
Specifically, we impose shear force to the fluid element such that
(i) the face OA moves with a velocity δv over a distance δx, and
(ii) the face OB moves with a velocity δu over a distance δy

Because of the velocity differential δv over a distance δx, the face *OA* rotates <u>counterclockwise</u> by an angle $\delta\alpha = \delta v \cdot \delta t / \delta x$ over a small period of time δt. Therefore the angular velocity of *OA* is

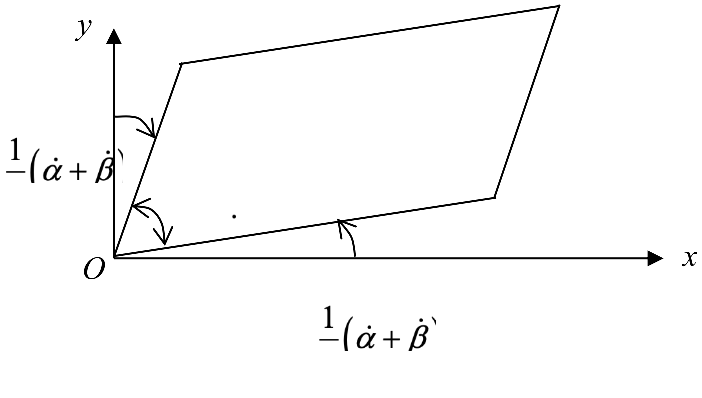
Similarly, the face *OB* rotates <u>clockwise</u> at an angular velocity given by



The deformation can be decomposed into a rigid body rotation at an angular velocity $\omega = \frac{1}{2}\left(\dot{\alpha} - \dot{\beta}\right) = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)$, where counterclockwise rotation is taken to be positive,



and an angular deformation, where the corner angle decreases at a rate given by $\dot{\gamma} = \dot{\alpha} + \dot{\beta} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}$, where a positive rate means a decreasing angle,



- Rate of angular deformation of a 2-D fluid element moving in the *x-y* plane (angular deformation is considered to be positive if it is to decrease the original right angle) is hence defined to be

(Optional) For a 3-D element in general, the rate of change of the corner angle that is initially a right angle between the *i-j* axes

$$\dot{\gamma}_{ij} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \quad (i \neq j) \qquad \text{or} \qquad \dot{\boldsymbol{\gamma}} = \nabla \boldsymbol{V} + (\nabla \boldsymbol{V})^T,$$

which is symmetric, i.e., $\dot{\gamma}_{ij} = \dot{\gamma}_{ji}$, and is called the deformation rate tensor.

- Rotation of a fluid element (about an axis which is perpendicular to the plane of the fluid motion) is the average of the angular velocities of the two mutually perpendicular sides of the element, where counterclockwise rotation is considered to be positive:

Rotation (or angular velocity) vector:

Note that for a 2-D flow in the *x-y* plane, ω_x and ω_y vanish identically; hence the rotation vector is always perpendicular to the *x-y* plane.

- To generalize, we may introduce:

(i) Strain rate tensor

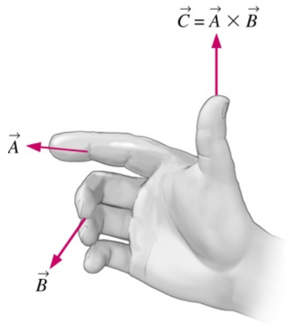
$$e_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) = \frac{1}{2}\text{deformation rate tensor} = \frac{1}{2}\dot{\gamma}_{ij}, \text{ or } \boldsymbol{e} = \frac{1}{2}\left[\nabla \boldsymbol{V} + (\nabla \boldsymbol{V})^T\right]$$

$$e_{ij} = \begin{cases} \text{normal strain rate} & i = j \\ \text{shear strain rate} & i \neq j \end{cases}$$

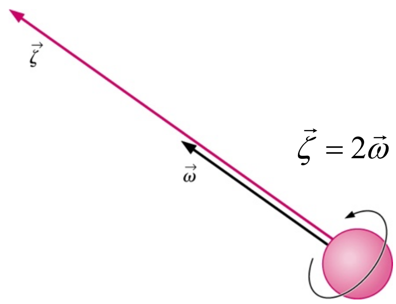
(ii) Vorticity vector = twice the rate of rotation, $\boldsymbol{\zeta} = 2\boldsymbol{\omega}$, where

vorticity $\boldsymbol{\zeta} = \nabla \times \boldsymbol{V}$ (curl of velocity)

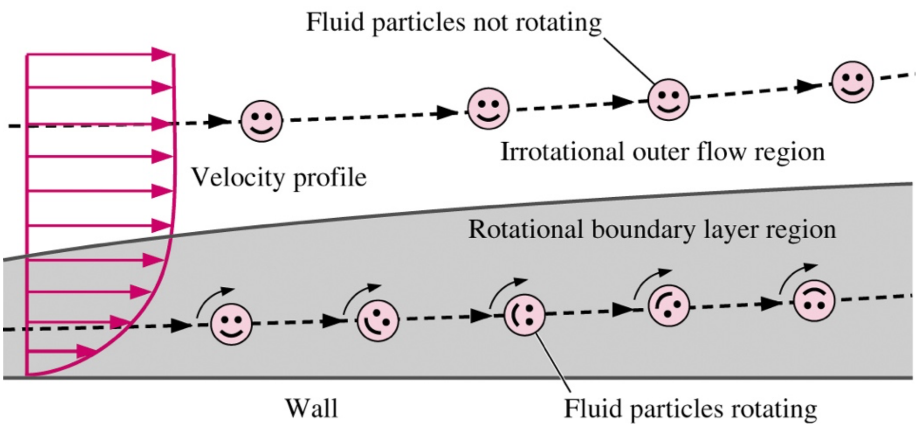
$$= \begin{vmatrix} \vec{i} & \vec{j} & \vec{k} \\ \dfrac{\partial}{\partial x} & \dfrac{\partial}{\partial y} & \dfrac{\partial}{\partial z} \\ u & v & w \end{vmatrix} = \underbrace{\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)}_{2\omega_x}\vec{i} + \underbrace{\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)}_{2\omega_y}\vec{j} + \underbrace{\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)}_{2\omega_z}\vec{k}$$



The direction of a vector cross product is determined by the right-hand rule. Therefore, the curl of velocity is always perpendicular to the velocity itself.

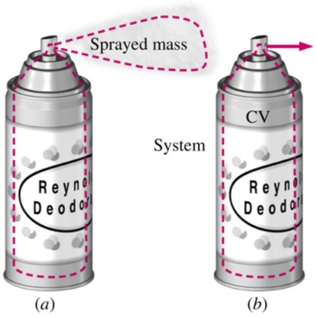


The vorticity vector is equal to twice the angular velocity vector of a rotating fluid particle.

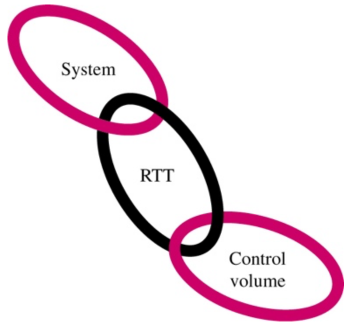


The difference between a rotational and irrotational flow: fluid elements in a rotational region of the flow rotate about their own axis, but those in an irrotational region of the flow do not.

C. The Reynolds Transport Theorem



Two approaches of analyzing a problem. (a) Lagrangian (System) approach: follow a collection of matter of the same identity as it moves and deforms; no mass crosses the boundary. (b) Eulerian (Control volume) approach: consider the changes in a certain fixed volume; mass crosses the boundary.



The *Reynolds transport theorem* (RTT) provides a link between the system approach and the control volume approach.

Define:

- Material Volume (MV): the volume that contains the collection of matter of the same identity; it may move and deform following the motion of the matter
- Material Surface: the enclosing surface of the material volume; by definition no matter can cross it.
- Control Volume (CV): a fixed volume in space; under motion matter which is initially in the CV may leave the CV at later times and new matter may get into the CV.
- Control Surface (CS): the enclosing surface of the CV.
- Flux: amount of matter (e.g., mass, momentum, energy) crossing a unit area of a surface per unit time.

We state without proof the Reynolds transport theorem, which provides a basis for developing differential equations for the various conservation laws:

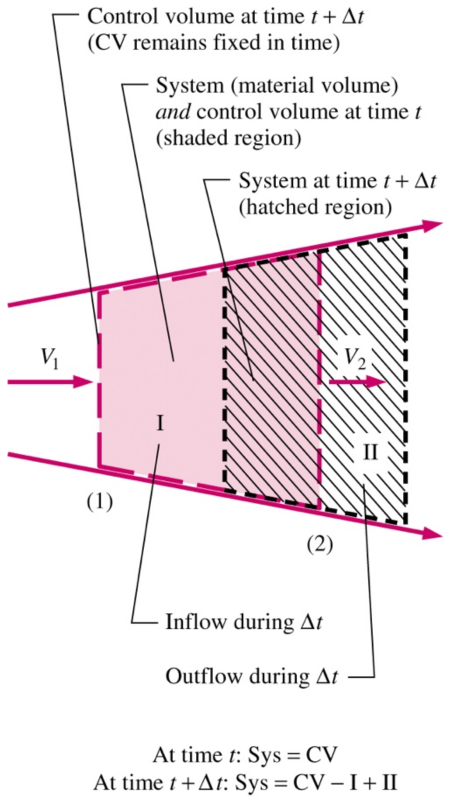
$$\underbrace{\frac{d}{dt}\iiint_{MV} \rho b \, d\forall}_{\text{rate of change of the property within the material volume}} = \underbrace{\frac{\partial}{\partial t}\iiint_{CV} \rho b \, d\forall}_{\text{local rate of change of the property within the fixed control volume that happens to coincide with the material volume at that instant}} + \underbrace{\iint_{CS} \rho b \mathbf{V} \cdot \mathbf{n} \, dA}_{\text{net out-flux of the property across the entire control surface}}$$

where ρ is density of fluid

b = B/m is an intensive version of an extensive property B (e.g. mass when b = 1, momentum when b = velocity **V**, kinetic energy when b = $V^2/2$)

m is mass

n is the unit vector outward normal to CS



A moving *system* (hatched region) and a fixed *control volume* (shaded region) in a diverging portion of a flow field at times t and $t + \Delta t$.

For b = 1, the property is mass and the left-hand side of the RTT becomes zero. Physically, this implies that there cannot be a rate of change of mass if we are tracking the same identity of mass --- mass cannot be created or destroyed! This left with the right-hand side of the RTT, which concerns only CV.

To illustrate the equation physically, let us set such a CV in a channel.

The CV has four CS, i.e. the upper, left, lower, and right face. Suppose there is a fluid flow in the system such that mass can flow in or out of the CV. In this example, mass can only flow in or out of the CV through the left and the right face. Let us denote the cross-sectional area of the left and the right face by A_1 and A_2, respectively; and denote the fluid velocity across the left and the right face by V_1 and V_2, respectively. With these notations, Eq. (1) can be written as

Note that ρAV is the mass flow (unit: kgs^{-1}). Densities of fluid on the left ρ_1 and right face ρ_2 are not the same in general. Physically, Eq. (2) implies that the rate of decrease of the mass in the CV is equal to the net rate of flow of mass across the CS.

In the specific case where the flow is steady (time-independent), the left-hand side of Eq. (2) becomes zero. Eq. (2) becomes,

This recovers what we studied in previous chapter, where the mass flowrates into and out of the CV equal.

D. Conservation of Mass

If the property is mass, then $b = 1$, and

L.H.S. $\quad \dfrac{d}{dt}\iiint_{MV} \rho d\cancel{V} = \dfrac{d}{dt}\left(\text{mass in } MV\right) = 0$

(by definition of MV, which always contains the same fluid)

R.H.S. $\quad \dfrac{\partial}{\partial t}\iiint_{CV} \rho d\cancel{V} + \iint_{CS} \rho \boldsymbol{V} \cdot \boldsymbol{n} dA$

$$= \underbrace{\iiint_{CV} \frac{\partial \rho}{\partial t} d\cancel{V}}_{CV \text{ is stationary}} + \underbrace{\iiint_{CV} \nabla \cdot \left(\rho \boldsymbol{V}\right) d\cancel{V}}_{\text{by Gauss theorem}}$$

Equating L.H.S. and R.H.S., and removing the volume integral since CV is arbitrary, we get the differential form of **Continuity Equation**

INCOMPRESSIBLE FLOW is defined as one in which the density of a fluid particle is invariant with time $\Leftrightarrow \dfrac{d\rho}{dt} = 0$, which implies

$$\nabla \cdot \boldsymbol{V} = 0$$

(ie, divergence of velocity is zero for incompressible flow)

In Cartesian coordinates, the continuity equation for incompressible flow reads

$$\boxed{\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0}$$

Note that a flow with constant density is always incompressible, but an incompressible flow does not necessarily have a constant density (e.g., flow in a stratified sea).

E. Applied Forces

- Body force due to gravity on a small fluid element = $\rho \boldsymbol{g} d\forall$
- Surface stress $\boldsymbol{s} = \boldsymbol{\tau} \cdot \boldsymbol{n}$, where $\boldsymbol{n}$ is the unit outward normal vector to the surface, and

$$\boldsymbol{\tau} = \begin{bmatrix} \tau_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \tau_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \tau_{zz} \end{bmatrix} = \tau_{ij} \qquad (i, j = 1, 2, 3)$$

are the stress components on an infinitesimal cubic fluid element.

τ_{ij} is a second order tensor, where
the **fi**rst index i denotes the **f**ace (on which the stress acts) being normal to x_i,
and the **s**econd index j denotes the **s**tress component being in the x_j direction.

In the textbook, the normal stress is denoted by σ_{ii} in order to distinguish it from the shear stress τ_{ij} $(i \neq j)$.
It can be shown that τ_{ij} is symmetric, ie, $\tau_{ij} = \tau_{ji}$. Therefore there are only 6 independent stress components.

F. Conservation of Linear Momentum

Apply Newton's second law of motion to a material volume of fluid:

$$\underbrace{\frac{d}{dt}\iiint_{MV}\rho \boldsymbol{V}\,d\forall}_{\text{rate of change of momentum}} = \underbrace{\iint_{MS}\boldsymbol{s}\,dA}_{\text{surface stress}} + \underbrace{\iiint_{MV}\rho \boldsymbol{g}\,d\forall}_{\text{body force}}$$

The L.H.S. can be converted, using the transport theorem and the continuity equation, into $\frac{d}{dt}\iiint_{MV}\rho \boldsymbol{V}d\forall = \iiint_{MV}\rho\frac{d\boldsymbol{V}}{dt}d\forall$.

The first term on the R.H.S. is $\iint_{MS}\boldsymbol{s}dA = \iint_{MS}\boldsymbol{\tau}\cdot\boldsymbol{n}dA = \iiint_{MV}\nabla\cdot\boldsymbol{\tau}d\forall$ on using Gauss theorem.

Plugging these terms back, and removing the volume integral since the volume is arbitrary, we get the differential form of momentum equation

The left hand term is a total derivative, which can be expanded into the Eulerian form:

By now, there are more unknowns than equations. To close the problem, we need to introduce CONSTITUTIVE (stress vs strain-rate) relations to relate the stress and the kinematics.

If the fluid is **Newtonian**, a <u>linear relationship between stress and strain rate</u> is followed

$$\tau_{ij} = \underbrace{-p\delta_{ij}}_{\text{Isotropic stress}} + \underbrace{\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)}_{\text{Deviatoric stress}}$$

where p is pressure (a minus sign because pressure is compressive, and stress is positive tension),

and δ_{ij} (identity or isotropic tensor) $= \begin{cases}1 & \text{for } i=j\\ 0 & \text{for } i\neq j\end{cases}$, $\quad\mu =$ dynamic viscosity coefficient

The divergence of the stress tensor is

$$\frac{\partial \tau_{ij}}{\partial x_j} = -\frac{\partial p}{\partial x_j}\delta_{ij} + \mu\left(\frac{\partial^2 u_i}{\partial x_j \partial x_j} + \cancel{\frac{\partial^2 u_j}{\partial x_j \partial x_i}}\right) \qquad \left(\because\ \frac{\partial u_j}{\partial x_j} = 0 \text{ by continuity}\right)$$

$$= -\frac{\partial p}{\partial x_i} + \mu\frac{\partial^2 u_i}{\partial x_j \partial x_j}$$

or $\quad \nabla\cdot\boldsymbol{\tau} = -\nabla p + \mu\nabla^2\boldsymbol{V}$

Finally, on substituting the above relationship, we obtain the Navier-Stokes equations

$$\rho\left(\frac{\partial \boldsymbol{V}}{\partial t}+\boldsymbol{V}\cdot\nabla\boldsymbol{V}\right)=-\nabla p+\rho\boldsymbol{g}+\mu\nabla^2\boldsymbol{V}$$

or in index form,

$$\rho\left(\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}\right)=-\frac{\partial p}{\partial x_i}+\rho g_i+\mu\frac{\partial^2 u_i}{\partial x_j^2}\quad (i,j=1,2,3,\ \text{summation over repeated index})$$

$$\text{or}\quad \underset{\text{(I)}}{\frac{\partial u_i}{\partial t}}+\underset{\text{(II)}}{u_j\frac{\partial u_i}{\partial x_j}}=\underset{\text{(III)}}{-\frac{1}{\rho}\frac{\partial p}{\partial x_i}}+\underset{\text{(IV)}}{g_i}+\underset{\text{(V)}}{\nu\frac{\partial^2 u_i}{\partial x_j^2}}\quad (\text{Here, } i \text{ is a free index, and } j \text{ is a dummy index})$$

where $\nu=\mu/\rho$ is the kinematic viscosity (don't confuse it with the velocity v).

Meanings of the five terms:-

(I) – local acceleration or unsteady term;
(II) – convective acceleration (inertia), nonlinear term of the equation;
(III) – pressure gradient;
(IV) – gravity;
(V) – viscous diffusion of momentum owing to molecular viscosity of the fluid.

Now, we have 4 equations (1 continuity + 3 components of momentum) for the four variables u_x, u_y, u_z, and p as functions of space and time (x,y,z,t). Note that it is the pressure gradient, rather than the pressure itself that drives the flow. The unsteady term vanishes when the flow is independent of time. The inertia terms vanish for strictly one-dimensional flow. The viscous terms are dissipative terms, without which the system becomes conservative.

The Equations of Motion for an Incompressible Newtonian Fluid

In Rectangular Coordinates (*x*, *y*, *z*)

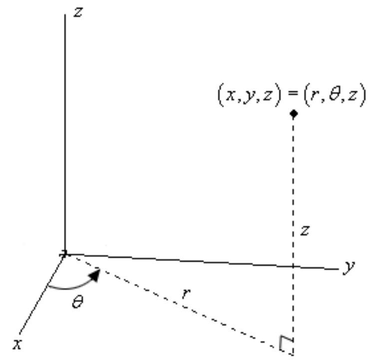
Continuity: $$\frac{\partial u_x}{\partial x}+\frac{\partial u_y}{\partial y}+\frac{\partial u_z}{\partial z}=0$$

x-component: $$\frac{\partial u_x}{\partial t}+u_x\frac{\partial u_x}{\partial x}+u_y\frac{\partial u_x}{\partial y}+u_z\frac{\partial u_x}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\nu\left(\frac{\partial^2 u_x}{\partial x^2}+\frac{\partial^2 u_x}{\partial y^2}+\frac{\partial^2 u_x}{\partial z^2}\right)+g_x$$

y-component: $$\frac{\partial u_y}{\partial t}+u_x\frac{\partial u_y}{\partial x}+u_y\frac{\partial u_y}{\partial y}+u_z\frac{\partial u_y}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\nu\left(\frac{\partial^2 u_y}{\partial x^2}+\frac{\partial^2 u_y}{\partial y^2}+\frac{\partial^2 u_y}{\partial z^2}\right)+g_y$$

z-component: $$\frac{\partial u_z}{\partial t}+u_x\frac{\partial u_z}{\partial x}+u_y\frac{\partial u_z}{\partial y}+u_z\frac{\partial u_z}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial z}+\nu\left(\frac{\partial^2 u_z}{\partial x^2}+\frac{\partial^2 u_z}{\partial y^2}+\frac{\partial^2 u_z}{\partial z^2}\right)+g_z$$

(g_x, g_y, g_z) are the components of the acceleration due to gravity in the x, y, and z directions. If, say, x and y are horizontal axes and z is positive upward, then $g_x = g_y = 0,$ and $g_z = -g$. Also, the gravity can be combined implicitly with the pressure term by introducing $p^* \equiv p - \rho \boldsymbol{g}\cdot\boldsymbol{x} = p - \rho\left(g_x x + g_y y + g_z z\right)$.



In Cylindrical Coordinates (*r*, *θ*, *z*)

Continuity: $$\frac{1}{r}\frac{\partial(ru_r)}{\partial r}+\frac{1}{r}\frac{\partial u_\theta}{\partial\theta}+\frac{\partial u_z}{\partial z}=0$$

r-component: $$\frac{\partial u_r}{\partial t}+u_r\frac{\partial u_r}{\partial r}+\frac{u_\theta}{r}\frac{\partial u_r}{\partial\theta}-\frac{u_\theta^2}{r}+u_z\frac{\partial u_r}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial r}$$
$$+\nu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}(ru_r)\right)+\frac{1}{r^2}\frac{\partial^2 u_r}{\partial\theta^2}-\frac{2}{r^2}\frac{\partial u_\theta}{\partial\theta}+\frac{\partial^2 u_r}{\partial z^2}\right]+g_r$$

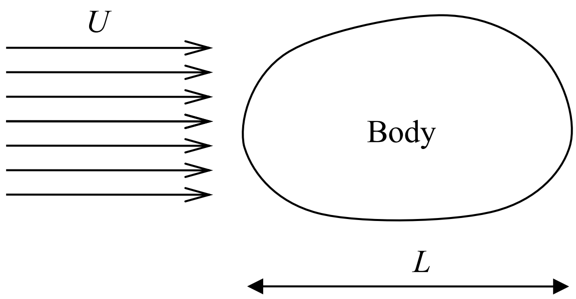
θ-component: $$\frac{\partial u_\theta}{\partial t}+u_r\frac{\partial u_\theta}{\partial r}+\frac{u_\theta}{r}\frac{\partial u_\theta}{\partial\theta}+\frac{u_r u_\theta}{r}+u_z\frac{\partial u_\theta}{\partial z}=-\frac{1}{\rho r}\frac{\partial p}{\partial\theta}$$
$$+\nu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}(ru_\theta)\right)+\frac{1}{r^2}\frac{\partial^2 u_\theta}{\partial\theta^2}+\frac{2}{r^2}\frac{\partial u_r}{\partial\theta}+\frac{\partial^2 u_\theta}{\partial z^2}\right]+g_\theta$$

z-component: $$\frac{\partial u_z}{\partial t}+u_r\frac{\partial u_z}{\partial r}+\frac{u_\theta}{r}\frac{\partial u_z}{\partial\theta}+u_z\frac{\partial u_z}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial z}$$
$$+\nu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_z}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 u_z}{\partial\theta^2}+\frac{\partial^2 u_z}{\partial z^2}\right]+g_z$$

G. Scaling and Approximation

- Because of the inertia terms (convective acceleration), the Navier-Stokes (NS) equations are non-linear equations.
- Except for simple flow geometry, analytical solutions do not exist in general.
- Fortunately, for many practical applications, not all terms in the equations are equally important, and therefore some subdominant terms can be dropped in favor of a first approximation of the problem. The approximate equations can then be solved (analytically or numerically) with much greater ease than the full-blown ones.
- It is important to judge, for a particular problem, the relative significance of the individual terms in the NS equations, which can be reflected from the magnitude of the corresponding non-dimensional parameters.

For illustration, consider incompressible unsteady flow past a body:



Characteristic scales:

Length (L); Time scale of unsteadiness (T); Velocity (U); Pressure (P)

Introduce dimensionless variables (distinguished by *):

$$\boldsymbol{V}^* = \boldsymbol{V}/U, \quad t^* = t/T, \quad \boldsymbol{x}^* = \boldsymbol{x}/L, \quad p^* = p/P, \quad \boldsymbol{g}^* = \boldsymbol{g}/g$$

the normalized Navier-Stokes equation can be expressed as

The scales have been chosen to be representative of the variables so that all the dimensionless terms are order unity. Now, the importance of each term (relative to the inertia) is carried by its bracketed coefficient.

$$\frac{L}{UT} = \text{Strouhal number (St)} = \frac{\text{temporal acceleration}}{\text{convective accelertion}}$$

$$\frac{P}{\rho U^2} = \text{Euler number (E)} = \frac{\text{pressure force}}{\text{inertia}}$$

$$\frac{UL}{\nu} = \text{Reynolds number (Re)} = \frac{\text{inerita}}{\text{viscous force}}$$

$$\frac{U^2}{gL} = \text{Froude number (Fr)} = \frac{\text{inerita}}{\text{gravity force}}$$

Possible Cases of Simplification:-

Large Re

Small St

Small Re

Spatial Dimension

Also, it is often the case that the flow varies only in one or two spatial dimensions, and therefore the problem can be reduced to a one- or two-dimensional problem, for which only one or two velocity components need to be solved. Some common cases of one-dimensional flow:

- fully developed pipe or channel flow: axial velocity as a function of radial distance from center of pipe $u = u(r)$, or longitudinal velocity as a function of distance from the bottom of channel $u = u(y)$;
- axi-symmetrical flow: velocity is symmetrical about an axis (e.g., point source/sink, vortex).

SIMPLE (EXACTLY OR NEARLY ONE-DIMENSIONAL) VISCOUS FLOW (BOOK CHAPTER 6, 8)

A. Mathematical Formulation for a Fluid Dynamics Problem

Assumptions:
- constant fluid properties (density ρ, viscosity μ)
- Newtonian fluid (linear, isotropic and purely viscous material)

Basic Variables:

Velocity $\boldsymbol{V} = (u, v, w) = \boldsymbol{V}(x, y, z, t)$ (3 variables)

Pressure $p = p(x, y, z, t)$ (1 variable)

Basic Governing Equations:

Continuity $\nabla \cdot \boldsymbol{V} = 0$ (1 equation)

Navier-Stokes $\dfrac{\partial \boldsymbol{V}}{\partial t} + \boldsymbol{V} \cdot \nabla \boldsymbol{V} = -\dfrac{1}{\rho}\nabla p + \boldsymbol{g} + \nu \nabla^2 \boldsymbol{V}$ (3 equations)

Other derived variables:

Stress $\boldsymbol{\tau}(x, y, z, t) = -p\boldsymbol{I} + \mu\left[\nabla \boldsymbol{V} + (\nabla \boldsymbol{V})^T\right]$, $\boldsymbol{I}$ = isotropic tensor

with stress components (see the definition on page 11):

$$\tau_{xx} = -p + 2\mu \frac{\partial u}{\partial x}, \quad \text{(normal stress)}$$

$$\tau_{yy} = -p + 2\mu \frac{\partial v}{\partial y}, \quad \text{(normal stress)}$$

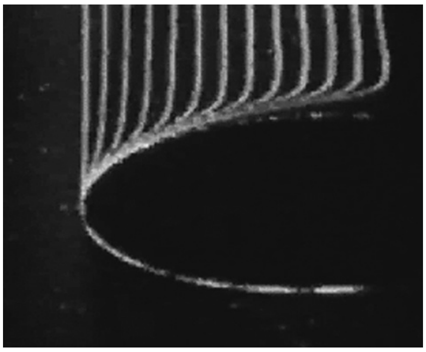
$$\tau_{xy} = \tau_{yx} = \mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \quad \text{(shear stress)}$$

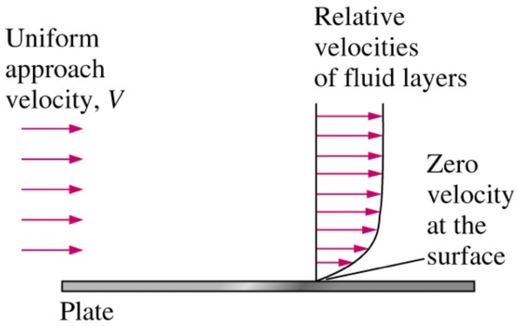
etc.

Vorticity $\boldsymbol{\zeta} = \nabla \times \boldsymbol{V} = \left(\dfrac{\partial w}{\partial y} - \dfrac{\partial v}{\partial z}\right)\vec{i} + \left(\dfrac{\partial u}{\partial z} - \dfrac{\partial w}{\partial x}\right)\vec{j} + \left(\dfrac{\partial v}{\partial x} - \dfrac{\partial u}{\partial y}\right)\vec{k}$

Boundary Conditions:
- No-slip boundary condition: the velocity of a fluid in contact with a solid impermeable wall must equal that of the wall

  If in particular the wall is stationary, the fluid adjacent to the wall must have zero velocity.

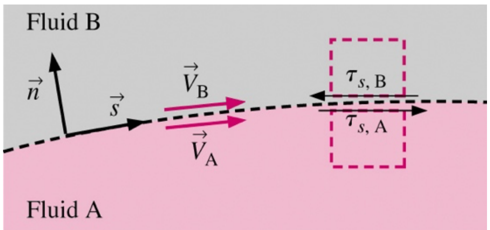




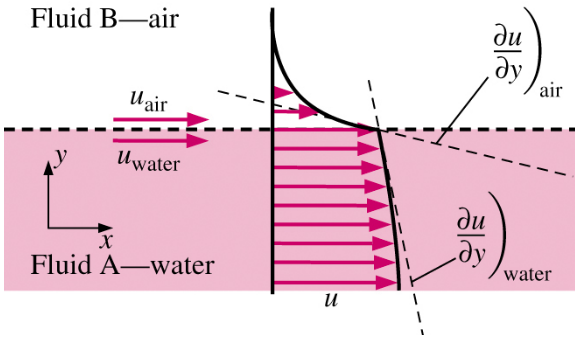
The development of velocity profiles due to the no-slip condition as a fluid flows past a blunt nose and a flat plate.

- Interface boundary condition between two fluids: when fluid A and fluid B meets at an interface, the velocity and stress must match between the two fluids at the interface

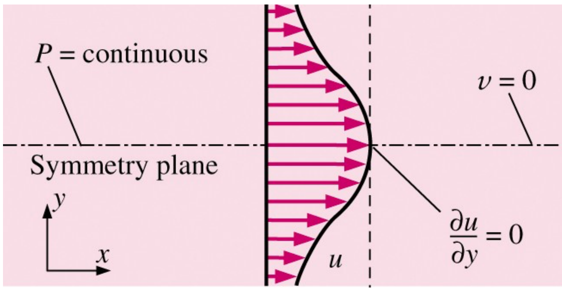
If, say, the interface is flat (along *x*-direction) and the fluids are moving parallel to the interface, the continuity of stress implies the continuity of pressure and shear stress at the interface



- Free-surface boundary condition: a degenerate form of the above interface boundary condition occurs at the *free-surface* of a liquid, meaning that fluid A is a liquid (say, water, oil) and fluid B is a gas (usually air). By virtue of the fact $\mu_{air} \ll \mu_{liquid}$, the shear stress at the air-liquid interface is negligibly small, and it is reasonable to approximate the shear stress to be at the interface, which is hence called a free surface,



- Other boundary conditions, such as inlet condition, outlet condition, periodic condition and symmetry, may also apply to certain types of boundaries, depending on the problem.



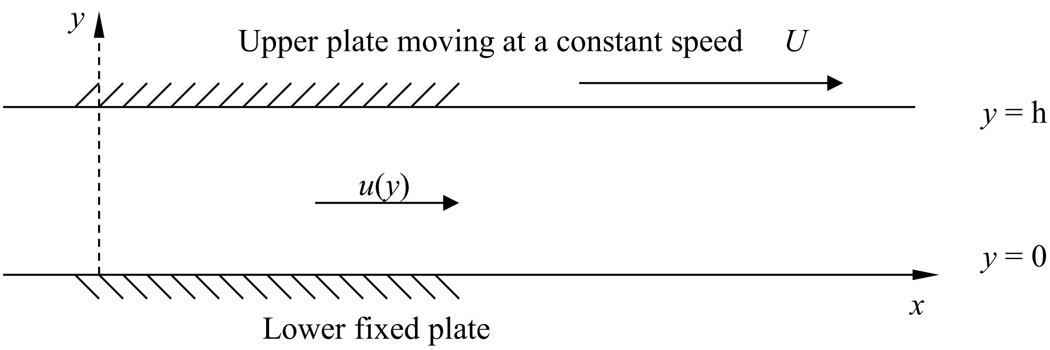
Boundary conditions along a plane of symmetry are defined so as to ensure that the flow field on one side of the symmetry plane is a mirror image of that on the other side, as shown above for a horizontal symmetry plane. The velocity gradient and the shear stress are zero on the plane of symmetry.

Initial Condition If the problem is time dependent (i.e., unsteady), an initial condition also needs to be specified.

****

Let us consider in the following sections a few applications of the Navier-Stokes equations, in which the flow configuration is simple enough for analytical solutions (exact or approximate) to be deduced. The assumptions are that the flow is steady ($\therefore\ \partial/\partial t = 0$), laminar, and incompressible and the fluid is Newtonian.

B. Plane Poiseuille-Couette Flow



Note that this is a unidirectional flow $u = u(y),\ \ v = 0$. Therefore there is no dependence on x for all variables: $\partial/\partial x = 0$.

The flow is driven by three forcings: (1) motion of the upper plate; (2) pressure gradient in the x-direction, $\partial p/\partial x = \text{a constant}$; (3) gravity, if x is not in a horizontal direction.

Recall the momentum equations:

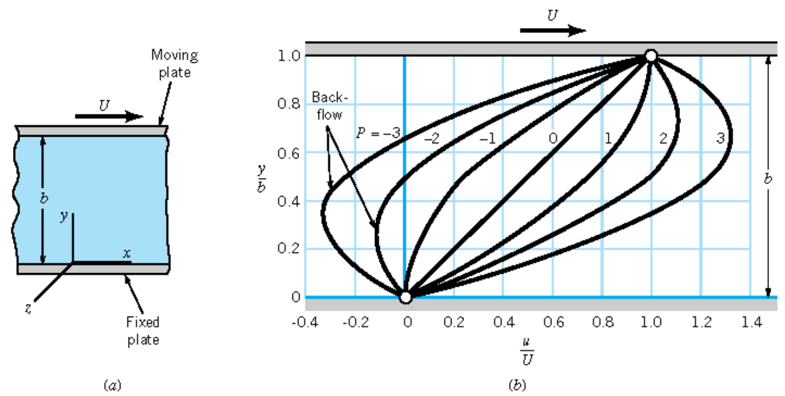
Note that the inertia terms are identically zero, which is true for all unidirectional flows irrespective of the Reynolds number.

Equation (2) simply gives that the pressure

The R.H.S. of equation (1) is constant, so the equation can be integrated twice with respect to y, giving

where C_1 and C_2 are integration constants that can be determined using the boundary conditions that

Solving for these constants, we obtain the solution for the velocity profile (see Fig. 6.31 below):



Couette flow is caused by the motion of a boundary wall moving in its own plane, while **Poiseuille flow** is caused by axial pressure gradient or gravity in the direction of flow.

The shear stress in the flow is

The discharge (flow-rate) per unit width of channel is given by

The volume flow averaged (mean) velocity

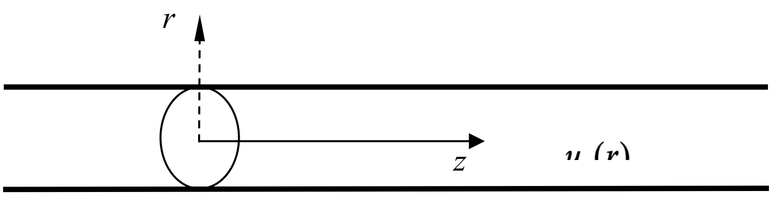
<u>It is left as an exercise for you to show the following</u>

> Given that $-\partial p / \partial x$ is a positive constant and $g_x = 0$, determine the location of the maximum velocity. It is also the point where the shear stress vanishes (why?). Hence, find the minimum value of U such that the shear stress will not vanish throughout the flow.

C. Circular Poiseuille Flow

We now consider laminar flow through a circular tube:

- The objective to find the relationship between volumetric flow rate and pressure change along a pipe of circular section.
- Examples include blood flow in capillaries, air flow in lung alveoli, where the Reynolds number is not high enough for the flow to become turbulent.
- Navier-Stokes equations in cylindrical coordinates are to be used, where $\partial / \partial \theta = 0$, since the flow is axially-symmetric (i.e., no dependence on angular position in a cross-section of the flow).
- We have seen that the gravity can be combined with the pressure gradient in a trivial manner, so let us ignore gravity in the following analysis.



Circular pipe of radius R

Again, this is a unidirectional flow: $u_r = u_\theta = 0$, $u_z \neq 0$ is driven by a constant and steady pressure gradient dp/dz in the axial direction.

The continuity equation reduces to

$$\cancel{\frac{1}{r}\frac{\partial(ru_r)}{\partial r}}+\cancel{\frac{1}{r}\frac{\partial u_\theta}{\partial\theta}}+\frac{\partial u_z}{\partial z}=0\Rightarrow u_z \text{ must not depend on } z, \qquad \therefore u_z=u_z(r)$$

The z-component momentum equation is simplified to

$$\cancel{\frac{\partial u_z}{\partial t}}+u_r\cancel{\frac{\partial u_z}{\partial r}}+\frac{u_\theta}{r}\cancel{\frac{\partial u_z}{\partial\theta}}+u_z\cancel{\frac{\partial u_z}{\partial z}}=-\frac{1}{\rho}\frac{\partial p}{\partial z}$$
$$+\nu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_z}{\partial r}\right)+\cancel{\frac{1}{r^2}\frac{\partial^2 u_z}{\partial\theta^2}}+\cancel{\frac{\partial^2 u_z}{\partial z^2}}\right]+\cancel{g_z}$$

$\Rightarrow \quad \dfrac{d}{dr}\left(r\dfrac{du_z}{dr}\right)=\dfrac{r}{\mu}\dfrac{dp}{dz}$, which can be integrated twice with respect to r to give

$$u_z(r)=\frac{r^2}{4\mu}\frac{dp}{dz}+C_1\ln r+C_2$$

The two integration constants C_1 and C_2 can be determined using the boundary conditions:

Plugging back, we get the expression for the velocity profile

$$u_z(r)=-\frac{dp}{dz}\frac{R^2}{4\mu}\left[1-\frac{r^2}{R^2}\right]$$

which is a parabolic distribution with the maximum at the center:

$$u_{\max}=u_z(r=0)=-\frac{R^2}{4\mu}\frac{dp}{dz}$$

The flow-rate is

$$Q=\int_A u_z dA=2\pi\int_0^R u_z r dr=-\frac{\pi R^4}{8\mu}\frac{dp}{dz}$$

The mean velocity is half the maximum velocity

$$\bar{u}=Q/A=\frac{-\frac{\pi R^4}{8\mu}\frac{dp}{dz}}{\pi R^2}=-\frac{R^2}{8\mu}\frac{dp}{dz}=\frac{u_{\max}}{2} \qquad \text{....................} \quad (1)$$

The shear stress at wall is given by

$$\tau_w=-\mu\left.\frac{du_z}{dr}\right|_{r=R}=-\frac{R}{2}\frac{dp}{dz}=4\mu\frac{\bar{u}}{R}$$

What is the purpose of considering the continuity equation?

Let's take the flow between two parallel plates as an example. It will lead us to $u = u(y)$ for simplifying the Navier-Stokes equation.

Thus, if you are given that $u = u(y)$ in the question, that implies that you do not need to consider/solve the continuity equation.

However, if you are given only $v = w = 0$, you need to apply this condition in the continuity equation, which will give you $\partial u/\partial x = 0$, implying that $u = u(y,z) = u(y)$. The last equality is due to the fact that the plates are infinite in the z-direction so we should not expect u to depend on z.

INVISCID AND POTENTIAL FLOW (BOOK CHAPTER 6)

Analysis can be considerably simplified if the flow under consideration can be regarded as INVISCID and IRROTATIONAL.

A. Inviscid (Nonviscous) Flow

- Flow of an ideal fluid with zero viscosity $(\mu = 0)$ would be inviscid exactly.
- In practice, flow is approximately inviscid when the effects of shear stresses on the motion are small as compared to other influences. One guiding condition is that the Reynolds number Re must be very large:

- Many flows involving water or air, whose viscosity is small, can practically be considered as inviscid as long as the viscous effects are not dominant (e.g., far from a wall).
- When the viscous force becomes negligible, the Navier-Stokes equations reduce to Euler's equations

- For incompressible flow, Euler's equations of motion can be integrated along a streamline to yield the Bernoulli equation

- It is remarkable that the Bernoulli equation provides an algebraic (rather than vector differential) relationship between pressure, velocity and position in the earth's gravitational field.

B. Irrotational (Potential) Flow

- Recall that vorticity (curl of velocity) is twice the rotation (angular velocity) of a fluid element.
- A fluid element will acquire vorticity when acted upon by a couple to cause it to rotate. One source of rotation is unbalanced shear stresses acting on its periphery. When shear stresses are absent, it is possible that the flow is irrotational.
- A flow field is **irrotational** if, at every point, the vorticity vanishes or

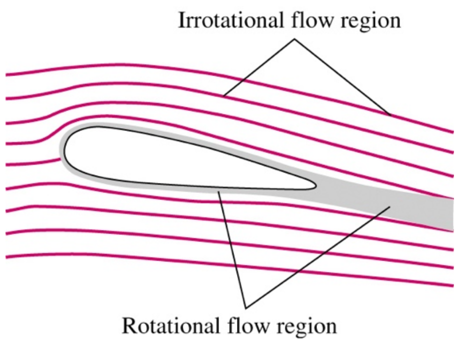
- It can be shown that the flow of an inviscid fluid which is irrotational at a particular instant of time remains irrotational for all subsequent times. That means, the motion of an inviscid fluid which is started from rest is always irrotational (provided the flow lies outside a boundary layer).
- This result is known as the **Persistence of Irrotational Motion** of an inviscid fluid. It is because the setting up of a rotation would require forces tangential to the boundary; and such forces, which arise through the viscous properties of the fluid, are non-existent in the inviscid fluid model.

- The constant in the Bernoulli equation becomes universal (i.e., not specific to a streamline) when the flow is irrotational (Section 6.4.4). Therefore, for incompressible irrotational flow, the Bernoulli equation can be applied between any two points in the flow field:

- The procedures of finding a solution for an irrotational flow field are typically:
    - Firstly, solve for the velocity field from an the governing <u>potential flow equation</u> derived from the condition of zero vorticity, which is the subject matter of the following sections.
    - Secondly, find the pressure from the Bernoulli equation.

  One should appreciate that solving irrotational flow equations is usually much simpler than solving the full Navier-Stokes equations. It is, however, important to note that the character of the governing equation has changed. The Navier-Stokes equations for viscous flow are second-order differential equations, while the Euler equation for inviscid flow involves only first-order derivatives of the velocity. This has important implications in terms of the number of boundary conditions necessary and possible to satisfy for a given problem. **For inviscid flow, it is not possible to satisfy the no-slip condition at a solid boundary.** In other words, on solving the Euler equation, we need not (or cannot) specify any condition on the tangential velocity of fluid in contact with a solid wall or another fluid. Slip on fluid–solid or fluid–fluid interfaces is freely allowed for ideal fluids.

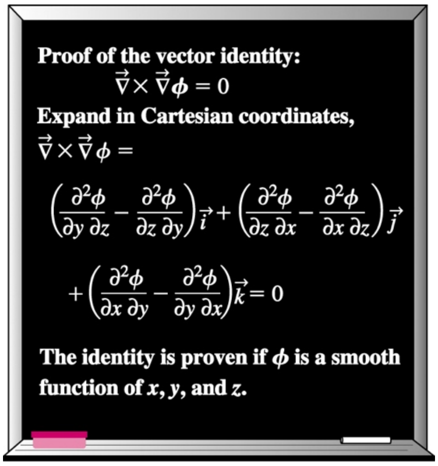
- No matter how small its viscosity is, a **real** fluid cannot freely "slide" past a solid boundary. Hence, irrotationality will **fail** to apply to a boundary layer, which is a thin layer that develops next to a solid wall owing to no-slip of the flow at the wall. The flow in a boundary layer is always viscous and highly rotational (a rapid change in velocity from zero at wall to the free stream value over a short distance); real fluid behavior must be accounted for in a boundary layer.



- In summary, the most general governing equation for a viscous or inviscid flow is the Navier-Stokes equation.
    - By setting the viscous term equals to zero, the scope is restricted to an inviscid flow [An inviscid flow is still rotational in general!]. The Navier-Stokes equation reduces to the Euler equation. In other words, the Euler equation is a special case of the Navier-Stokes equation.
    - By setting the viscous term AND vorticity equal to zero, the scope is restricted to an irrotational flow. The Navier-Stokes equation reduces to the potential flow equation, which we will discuss next. In other words, the potential flow equation is a special case of the Euler equation, which is a special case of Navier-Stokes equation.

C. The Velocity Potential

- For any scalar field ϕ, curl(grad ϕ) = **0** is an identity. See a proof below.



- Alternatively speaking, a velocity field $\boldsymbol{V}$ is irrotational or curl $\boldsymbol{V}$ = **0** if and only if there exists a scalar field ϕ such that $\boldsymbol{V}$ = grad ϕ.

- The scalar function is called **velocity potential**

$$\boldsymbol{V} \equiv \nabla \phi$$

Cartesian coordinates: $$u = \frac{\partial \phi}{\partial x}, \qquad v = \frac{\partial \phi}{\partial y}, \qquad w = \frac{\partial \phi}{\partial z}$$

Cylindrical coordinates: $$u_r = \frac{\partial \phi}{\partial r}, \qquad u_\theta = \frac{1}{r}\frac{\partial \phi}{\partial \theta}, \qquad u_z = \frac{\partial \phi}{\partial z}$$

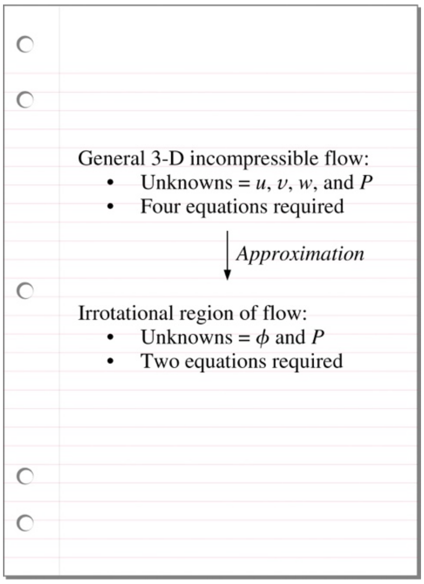
Irrotational flow is therefore also called **potential flow**.

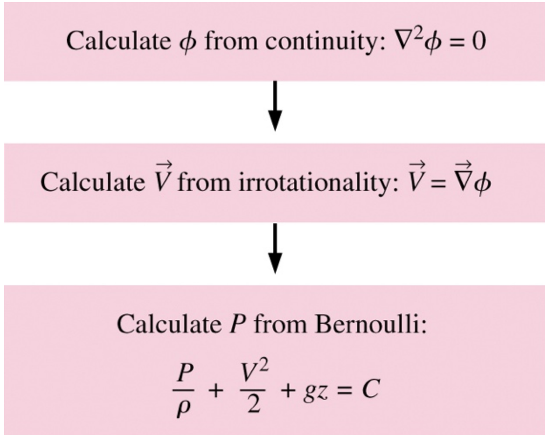
- The velocity potential satisfies **Laplace's equation** on substituting the above relation into the continuity equation:

$$\nabla \bullet \boldsymbol{V} = 0 \Rightarrow \nabla \bullet \nabla \phi = 0, \quad \text{or} \quad \nabla^2 \phi = 0$$

Cartesian coordinates: $$\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = 0$$

Cylindrical coordinates: $$\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial \phi}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 \phi}{\partial \theta^2} + \frac{\partial^2 \phi}{\partial z^2} = 0$$



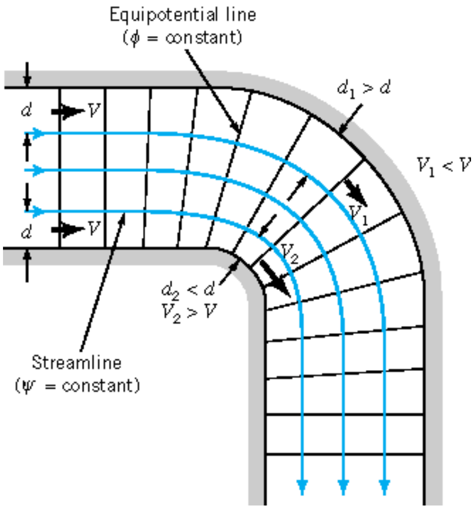


D. Equipotential Lines and Streamlines

- A *two-dimensional* potential flow field can be graphically represented using a **flow net** composed of equipotential lines and streamlines.

- Equipotential lines are (contour) lines of constant velocity potential, while streamlines are lines in the flow field that are everywhere tangent to the velocity. It can be shown that these two sets of lines are orthogonal (i.e., they intersect each other at right angles).

Figure 6.15 shows a flow net for a 90° bend. A flow net is useful in the visualization of a flow pattern. To further understand what information a flow net can provide, we need to know something about stream function.



Stream Function

- For 2-D incompressible flow, another scalar function, viz **stream function** can be introduced to identically satisfy the continuity equation.

  A stream function $\psi(x,y)$ or $\psi(r,\theta)$ is defined such that

  $u = \dfrac{\partial \psi}{\partial y}, \quad v = -\dfrac{\partial \psi}{\partial x}$ for 2-D flow in Cartesian coordinates,

  which satisfies $\dfrac{\partial u}{\partial x} + \dfrac{\partial v}{\partial y} = 0$ identically.

  $u_r = \dfrac{1}{r}\dfrac{\partial \psi}{\partial \theta}, \quad u_\theta = -\dfrac{\partial \psi}{\partial r}$ for 2-D flow in Polar coordinates,

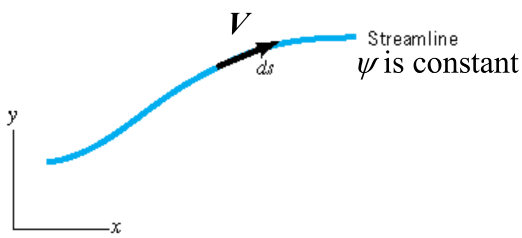
  which satisfies $\dfrac{\partial}{\partial r}(ru_r) + \dfrac{\partial u_\theta}{\partial \theta} = 0$ identically.

  Note that the stream function is introduced based on *kinematics* consideration only. It is definable for any two-dimensional incompressible flow fields, irrespective of the flow being inviscid or not.

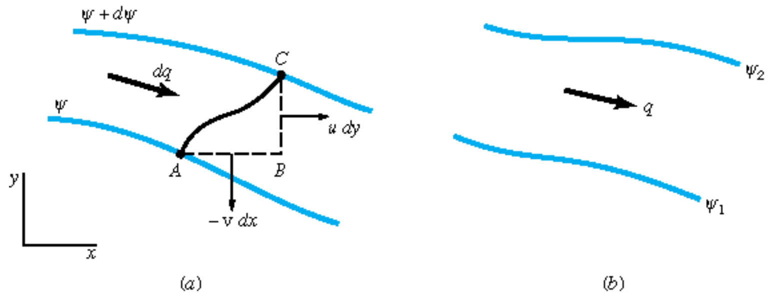
- Physically, ψ **is constant along a streamline** since

  That means, a line of constant ψ (along which $d\psi = 0$) will have its slope in the same direction of flow: $dy/dx = v/u$. This is nothing but the defining property for a streamline.

  Note that a solid boundary is always a streamline. At a particular instant of time, there is no fluid crossing any streamline, and distinct streamlines cannot cross.



- Given any two points in space whose stream function values are known, then the volume flow rate across any line joining these two points is equal to the difference in values of their stream functions.



- If the 2-D flow is irrotational, the stream function also satisfies Laplace's equation, since

Therefore, for two-dimensional irrotational flow, both the velocity potential ϕ and the stream function ψ satisfy Laplace's equation. They are called **harmonic functions**, and they are **harmonic conjugates** of each other. These functions are related, but their origins are different:

– The stream function is defined by continuity; the Laplace equation for ψ results from irrotationality.

– The velocity potential is defined by irrotationality; the Laplace equation for ϕ results from continuity.

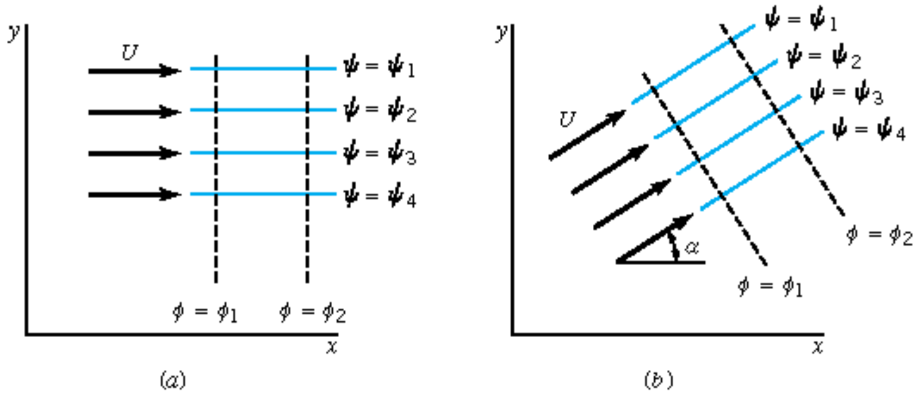
By now, referring back to Figure 6.15, you should understand that in a flow net the velocity is roughly given by

$$V \approx \frac{\Delta\phi}{\Delta n} \approx \frac{\Delta\psi}{\Delta s}$$

where Δn is the spacing between two adjacent equipotential lines, and Δs is the spacing between two adjacent streamlines. Therefore, the velocity is higher in a region where the mesh is finer, and lower where the mesh is coarser.

E. Some Simple Plane Potential Flows

1) Uniform Flow with constant velocity U

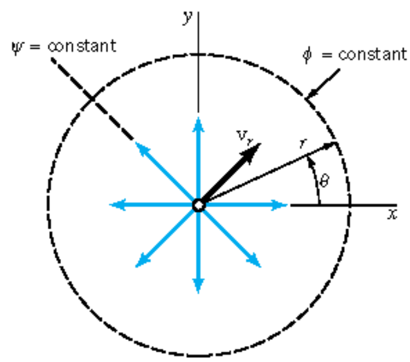


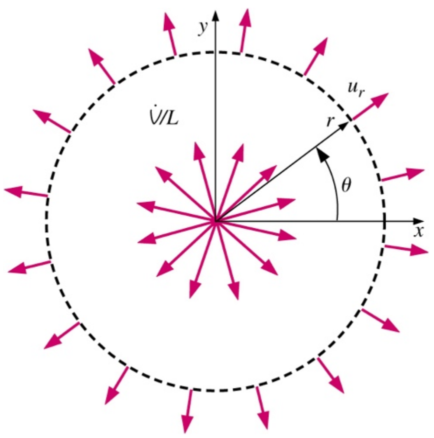
For case (a) where the flow is purely in the x-direction:

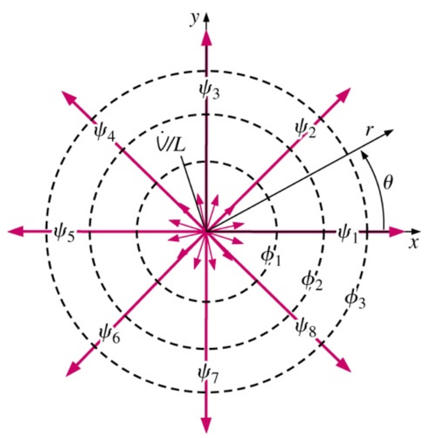
Can you write down the corresponding ϕ and ψ for case (b) where the flow is at an angle α with the x-axis?

2) Source and Sink

A 2-D source is a line (from a mathematical perspective) that runs perpendicular to the plane of flow and injects fluid equally in all directions. The figure shows the flow field of a source at the origin, from which fluid particles emerge and follow radial pathlines. The **strength** of a source, denoted by m, is the volume rate of flow emanating from unit length of the line.







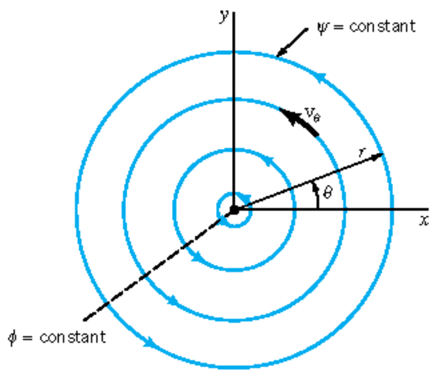
By conservation of mass, $m = 2\pi r u_r$ for any radial distance *r* from the source located at the origin. Hence, $u_r = \dfrac{\partial \phi}{\partial r} = \dfrac{1}{r}\dfrac{\partial \psi}{\partial \theta} = \dfrac{m}{2\pi r}$, $\quad u_\theta = \dfrac{1}{r}\dfrac{\partial \phi}{\partial \theta} = -\dfrac{\partial \psi}{\partial r} = 0$. On integrating,

- when $m > 0$, the flow is radially outward, the origin is a **SOURCE**
- when $m < 0$, the flow is radially inward, the origin is a **SINK**
- the origin is a singularity where $u_r \to \infty$
- conservation of mass is satisfied everywhere except the origin

3) <u>Vortex</u>

In contrast to a source, a vortex has the pathlines being circles centered on the origin, and fluid particles move along these circles. The vortex can be used to model the flow round the plughole in a bathtub. An irrotational vortex is called a free vortex. The strength of a vortex is measured by the *circulation* $\Gamma = \oint_C \boldsymbol{V} \cdot \boldsymbol{t} ds$ around a closed curve *C* that encloses the center of the vortex. Hence,

$$u_r = \frac{\partial \phi}{\partial r} = \frac{1}{r}\frac{\partial \psi}{\partial \theta} = 0, \quad u_\theta = \frac{1}{r}\frac{\partial \phi}{\partial \theta} = -\frac{\partial \psi}{\partial r} = \frac{\Gamma}{2\pi r}.$$



On integrating,

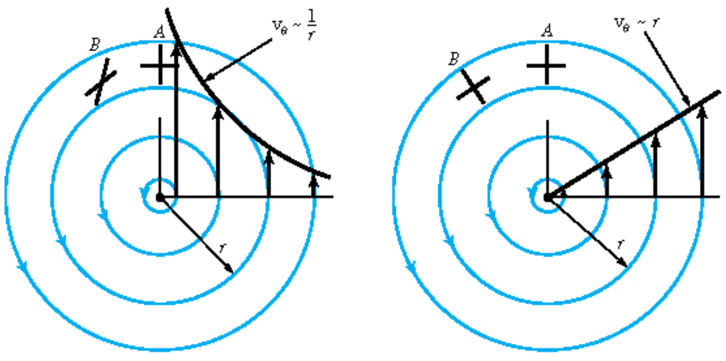
Velocity potential $\quad \phi = \dfrac{\Gamma}{2\pi}\theta$

($\therefore$ equipotential lines are radial lines)

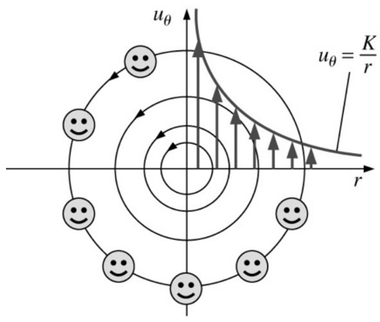
Stream function $\quad \psi = -\dfrac{\Gamma}{2\pi}\ln r$

($\therefore$ streamlines are concentric circles centered on the origin)

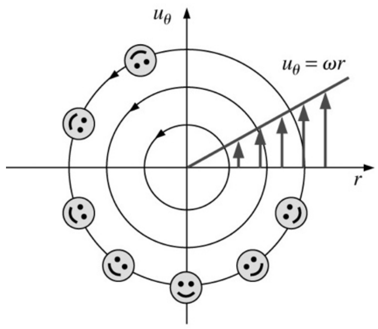
- the flow is not defined at the origin
- the vorticity curl $\boldsymbol{V} = 0$, except at $r = 0$, where $\boldsymbol{V}$ is not defined
- *free* vortex (a) is irrotational flow, tangential velocity decreases radially $u_\theta \propto r^{-1}$
- *forced* vortex (b) is rotational flow, tangential velocity increases radially $u_\theta \propto r$



(a) (b)



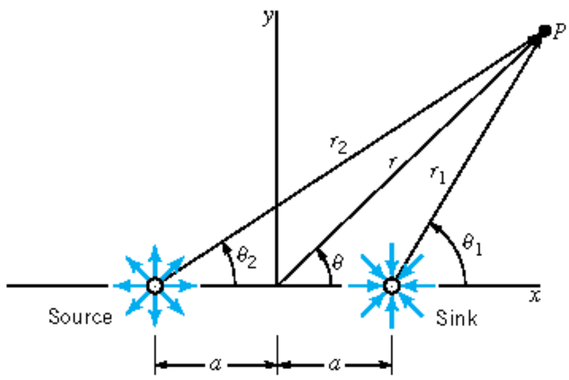
(a)

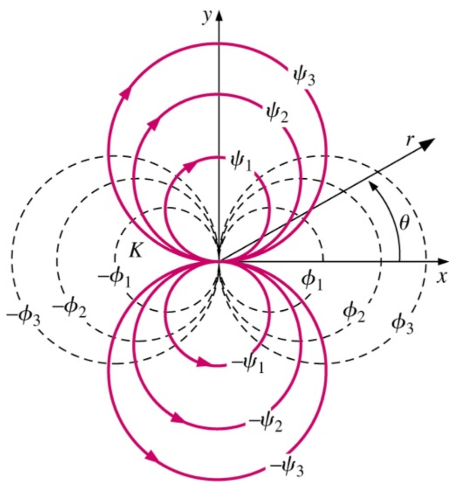


(b)

4) <u>Doublet</u>

Consider a combination of a source and a sink of equal strength m and separated at a distance $2a$ (left figure):





If the source and sink are moved indefinitely closer together $(a \to 0)$ in such a way that the product $2am$ (distance apart $\times$ strength) is kept finite and constant, then we obtain a **doublet**. The streamline pattern for a doublet is shown in the right figure above. The line joining the source to the sink is called the axis of the doublet, and is taken to be positive in the direction from sink to source. The **strength** of the doublet is

The basic potential flows that have been discussed so far are more mathematical constructions than physically realistic entities (although a source/sink may represent the flow field of an injection/withdrawing well, and so on). However a combination of these basic potential flows may provide a representation of some flow fields of practical interest. This is the subject matter for the next section.

TABLE 6.1
Summary of Basic, Plane Potential Flows.

Description of Flow Field	Velocity Potential	Stream Function	Velocity Components[a]
Uniform flow at angle α with the x axis (see Fig. 6.16b)	$\phi = U(x\cos\alpha + y\sin\alpha)$	$\psi = U(y\cos\alpha - x\sin\alpha)$	$u = U\cos\alpha$ $v = U\sin\alpha$
Source or sink (see Fig. 6.17) $m > 0$ source $m < 0$ sink	$\phi = \dfrac{m}{2\pi}\ln r$	$\psi = \dfrac{m}{2\pi}\theta$	$v_r = \dfrac{m}{2\pi r}$ $v_\theta = 0$
Free vortex (see Fig. 6.18) $\Gamma > 0$ counterclockwise motion $\Gamma < 0$ clockwise motion	$\phi = \dfrac{\Gamma}{2\pi}\theta$	$\psi = -\dfrac{\Gamma}{2\pi}\ln r$	$v_r = 0$ $v_\theta = \dfrac{\Gamma}{2\pi r}$
Doublet (see Fig. 6.23)	$\phi = \dfrac{K\cos\theta}{r}$	$\psi = -\dfrac{K\sin\theta}{r}$	$v_r = -\dfrac{K\cos\theta}{r^2}$ $v_\theta = -\dfrac{K\sin\theta}{r^2}$

[a]Velocity components are related to the velocity potential and stream function through the relationships:

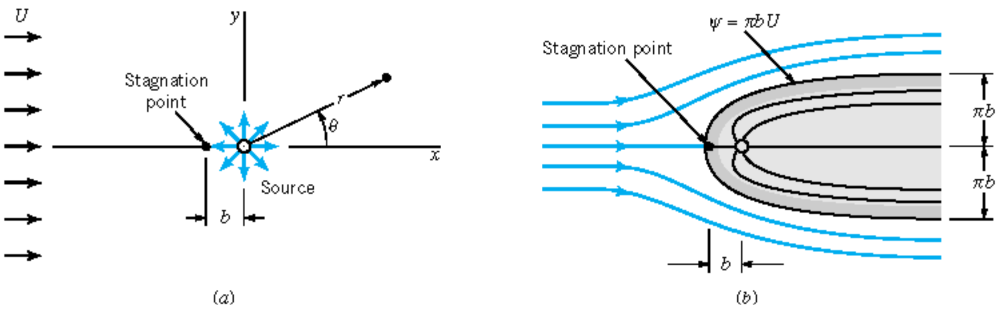
$$u = \frac{\partial\phi}{\partial x} = \frac{\partial\psi}{\partial y} \qquad v = \frac{\partial\phi}{\partial y} = -\frac{\partial\psi}{\partial x} \qquad v_r = \frac{\partial\phi}{\partial r} = \frac{1}{r}\frac{\partial\psi}{\partial\theta} \qquad v_\theta = \frac{1}{r}\frac{\partial\phi}{\partial\theta} = -\frac{\partial\psi}{\partial r}$$

F. Superposition of Basic Potential Flows

Let us first be reminded that for inviscid flow, a solid boundary is a streamline, and conversely, **a streamline can be considered as a solid boundary**. The kinematic conditions along the two are the same: normal velocity = 0. In fact, we may replace any streamline in a flow field by an impermeable surface without disturbing the flow.

Since the governing equation (Laplace's equation) for potential flow is linear, superposition of solutions gives the solution to the combined effect. In the following examples, you will see how **ideal flows** can be described by a combination of basic solutions. A key quantity to be determined is the **dividing streamline**. In the followings, we use two examples to illustrate the general procedures.

1) Source + Uniform Flow = Flow Past a Half Body



Step 1: Write down the stream function of the flow.

Referring to Table 6.1 on page 10, the stream function of the flow is given by

(1)

Step 2: Locate the stagnation point and obtain an expression for the stream function at the stagnation point.

It is apparent that the stagnation point is at the tip of the half body, at $x = -b$, as shown in the figure above. Considering the velocity component due to the uniform flow, which goes from left to right on the left-hand side of the stagnation point, this implies $u|_{\theta=0} = U\cos\theta|_{\theta=0} = U$. Considering the radial velocity component due to the source, which goes from right to left on the right-hand side of the stagnation point, this implies $u_r|_{r=b,(\theta=\pi)} = m/(2\pi r)|_{r=b,(\theta=\pi)} = m/(2\pi b)$. Note the different definition of θ in the uniform flow and the source, see page 7. At the stagnation point, these two flow components should equal, giving

(2)

An expression for the stream function at the stagnation point can be obtained by substituting (i) $\theta = 0$ into Eq. (1) for the uniform flow component and (ii) $\theta = \pi$ and $r = b$ into Eq. (1) for the source component. This gives

(3)

where Eq. (2) has been used to obtain the last equality in Eq. (3).

Step 3: Obtain an alternative expression for the stream function at the stagnation point and equate it to the previous one that we obtained.

An alternative expression for the stream function at the stagnation point can be obtained by substituting Eq. (2) into Eq. (1). This gives

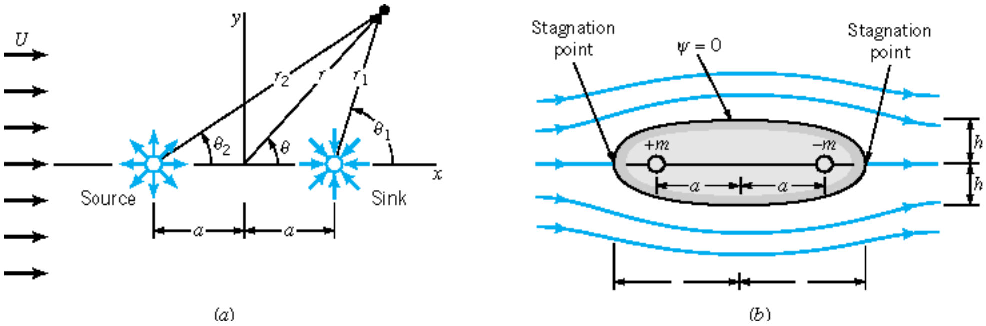
(4)

Equating Eq. (4) and Eq. (3) gives the dividing streamline as,

(5)

The flow pattern around the half-body is described by streamlines $|\psi| > \psi_{\text{stagnation}}$. The velocity components and the pressure can then be determined as described in earlier sections.

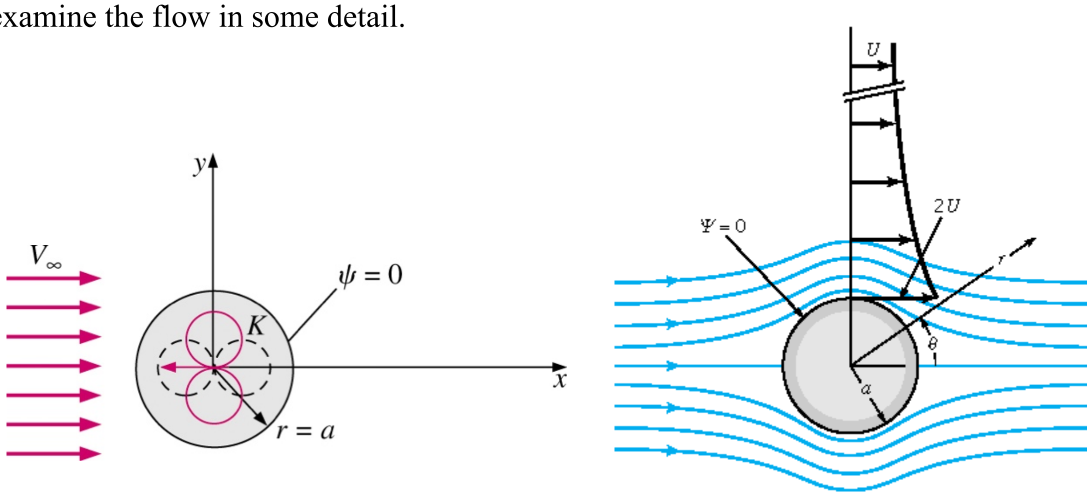
2) Source + Sink + Uniform Flow = Flow Past a Rankine Oval



The source and the sink are of the same strength: any mass of fluid injected by the source is eventually drawn into the sink. The dividing streamline is now a closed curve. This finite body, called Rankine Oval, has two stagnation points, one at the front end and the other at the rear end of its boundary.

3) <u>Doublet + Uniform Flow = Flow Past a Circular Cylinder</u>

As the source and the sink combine to become a doublet, the Rankine Oval becomes a circular cylinder. As the flow past a circular cylinder is of fundamental interest, let us examine the flow in some detail.



Step 1: Write down the stream function of the flow.

Referring to Table 6.1 on page 10, the stream function of the flow is given by

(1)

Step 2: Locate the stagnation point and obtain an expression for the stream function at the stagnation point.

There are two stagnation points in the flow. Let us pretend that we do not know about this and only observe the apparent stagnation point at the left tip of the cylinder, at $x = -a$, as shown in the figure above. Considering the velocity component due to the uniform flow, which goes from left to right on the left-hand side of the stagnation point, this implies $u|_{\theta=0} = U\cos\theta|_{\theta=0} = U$. Considering the radial velocity component due to the doublet, which goes from right to left on the right-hand side of the stagnation point, this implies $u_r|_{r=a,\theta=\pi} = -K\cos\theta/(r^2)|_{r=a,\theta=\pi} = K/(a^2)$. At the stagnation point, these two flow components should equal, giving

(2)

An expression for the stream function at the stagnation point can be obtained by substituting (i) $\theta = 0$ into Eq. (1) for the uniform flow component and (ii) $\theta = \pi$ and $r = a$ into Eq. (1) for the doublet component. This gives

(3)

Step 3: Obtain an alternative expression for the stream function at the stagnation point and equate it to the previous one that we obtained.

An alternative expression for the stream function at the stagnation point can be obtained by substituting Eq. (2) into Eq. (1). This gives

(4)

Equating Eq. (4) and Eq. (3) gives the dividing streamline as,

(5)

Using the above results, the stream function can be rewritten as

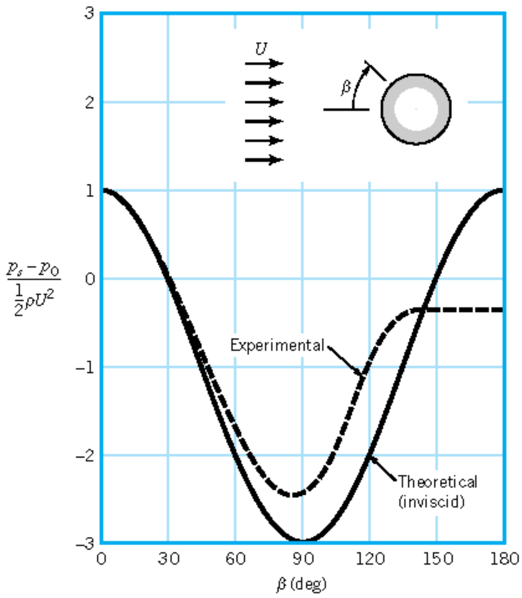
(6)

from which we obtain the velocity components

(7)

On the cylinder surface $r = a$, the tangential velocity is $u_\theta = -2U \sin\theta$. This analysis also indicates that there are in fact two stagnation points at $\theta = 0$ and $\theta = \pi$.

The pressure distribution on the cylinder surface can be found from the Bernoulli equation

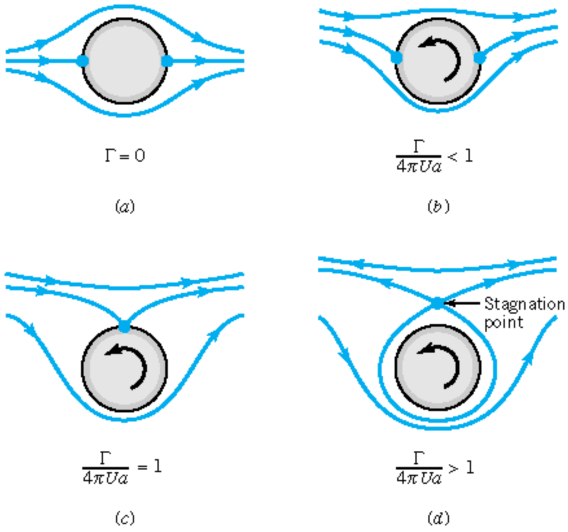


where p_0 is the far upstream pressure. It is remarkable that the pressure distribution is symmetrical about the horizontal and the vertical diameters. Therefore there is no net force arising from the pressure distribution around the cylinder in both streamwise and lateral directions. In other words, both drag and lift forces are exactly zero, as predicted from the potential flow theory.

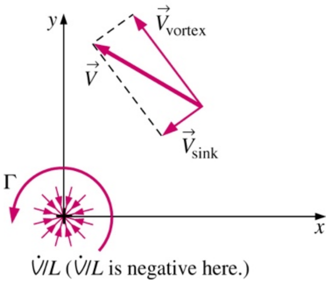
This zero drag prediction is contrary to what has been observed in reality. There is always a significant drag developed on a cylinder when it is placed in a stream of moving fluid. This discrepancy is called **d'Alembert's Paradox**, which was not explained until the concepts of **boundary layer** and **flow separation** were developed. A comparison between the inviscid and the real pressure distributions is shown above.

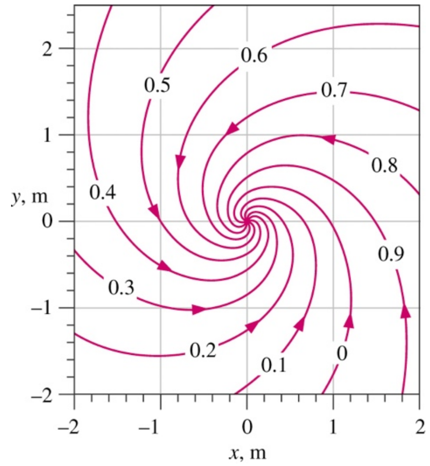
4) Free Vortex + Doublet + Uniform Flow = Flow Past a Rotating Circular Cylinder

The effect of adding a vortex is to upset the symmetry of flow about the horizontal diameter. Therefore, the pressure in the upper half of the cylinder is not balanced by the pressure in the lower half. This results in a net lift force acting laterally on the cylinder.

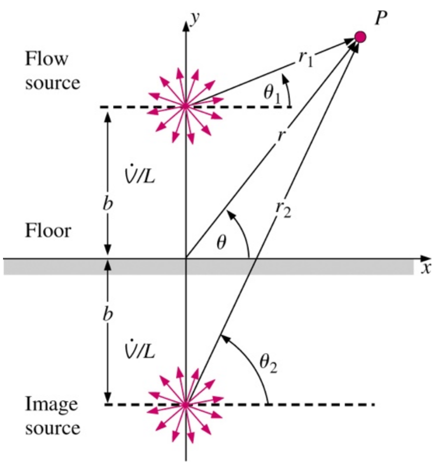


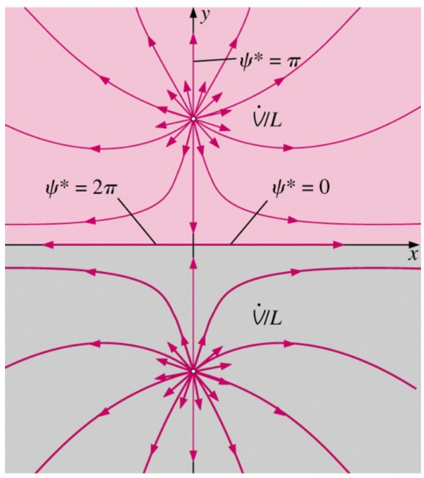
5) Sink + Free Vortex = Spiral Flow





6) Two separated sources of equal strength = source flow with a neighboring wall

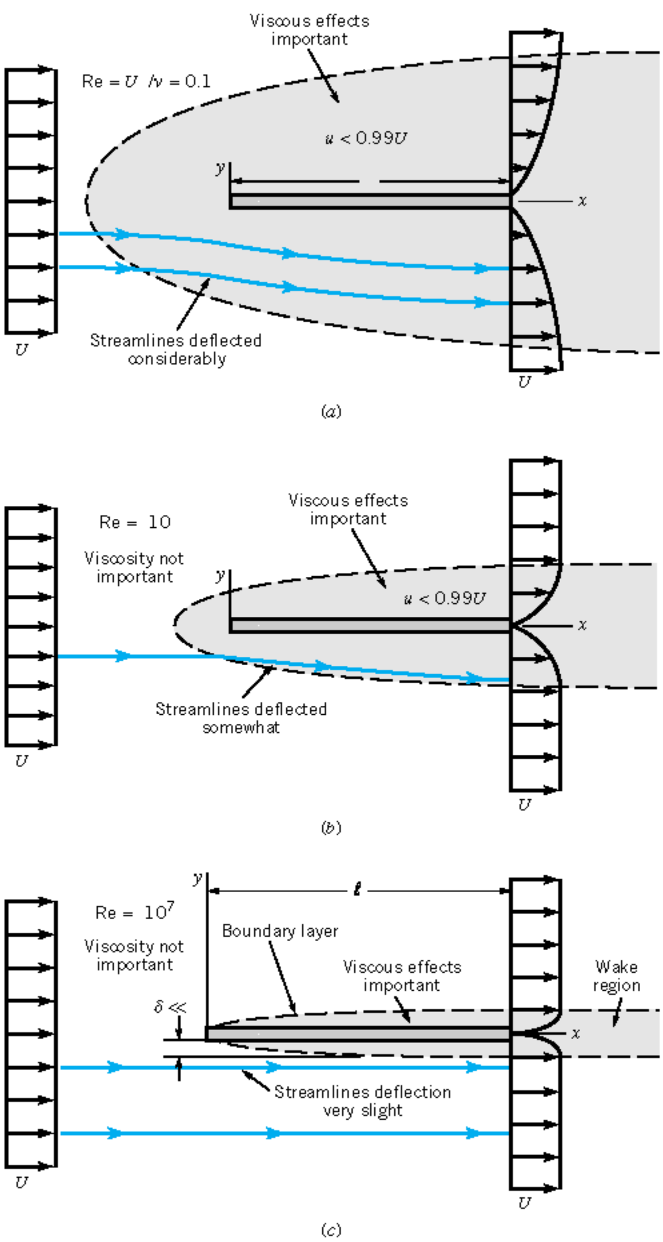




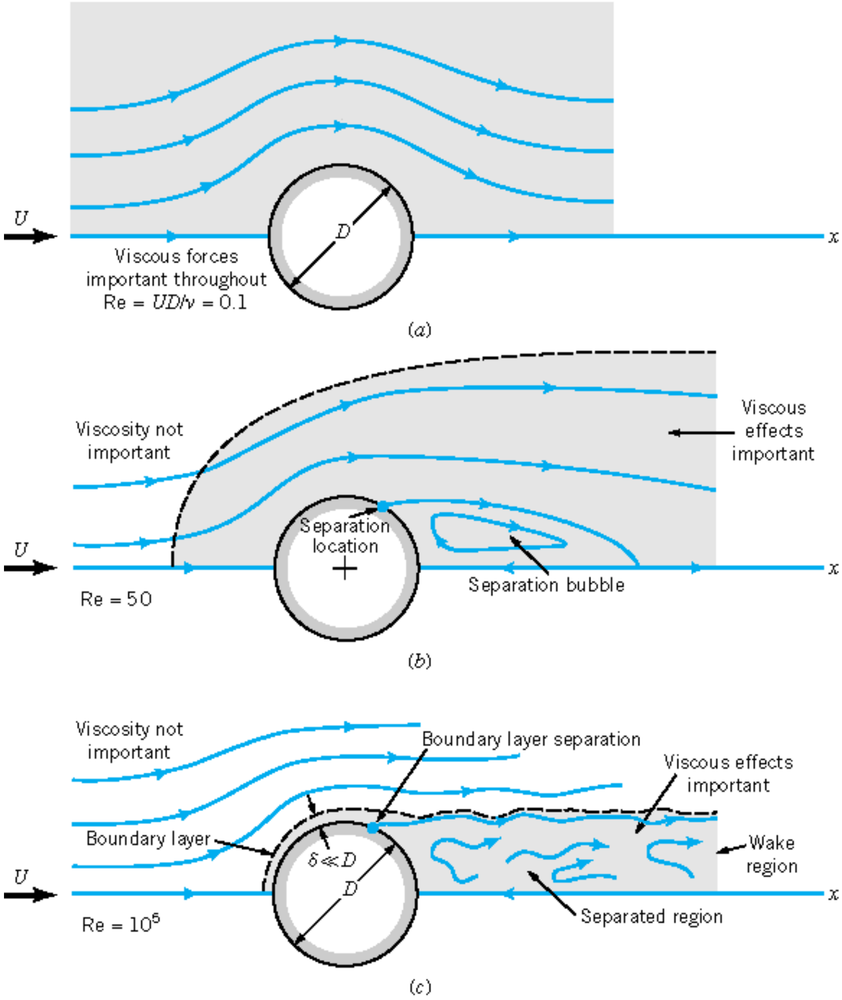
FLOW PAST A BODY AND BOUNDARY LAYER THEORY (BOOK CHAPTER 9)

A. Introduction

In 1904, Prandtl developed the concept of the boundary layer, which provides an important link between ideal-fluid flow (inviscid irrotational flow) and real-fluid flow (viscous rotational flow). It was accepted that for fluids with relatively small viscosity (or more exactly, flow with a high Reynolds number), the effect of internal friction in the fluid is appreciable only in a narrow region surrounding the fluid boundaries. Therefore the flow sufficiently far away from the solid boundaries may be considered as ideal flow (in which effects of viscosity are neglected). However, flow near the boundaries suffers retardation by the boundary shear forces and at the boundaries the velocity is zero (**no-slip** condition). A steep velocity gradient is therefore resulted in a thin layer adjacent to the boundaries, which is known as the **boundary layer**. It is of great significance when behavior of real fluid is considered. For example, it explains the **d'Alembert's paradox** – the drag force experienced by a cylinder in stream that cannot be predicted with a potential theory.



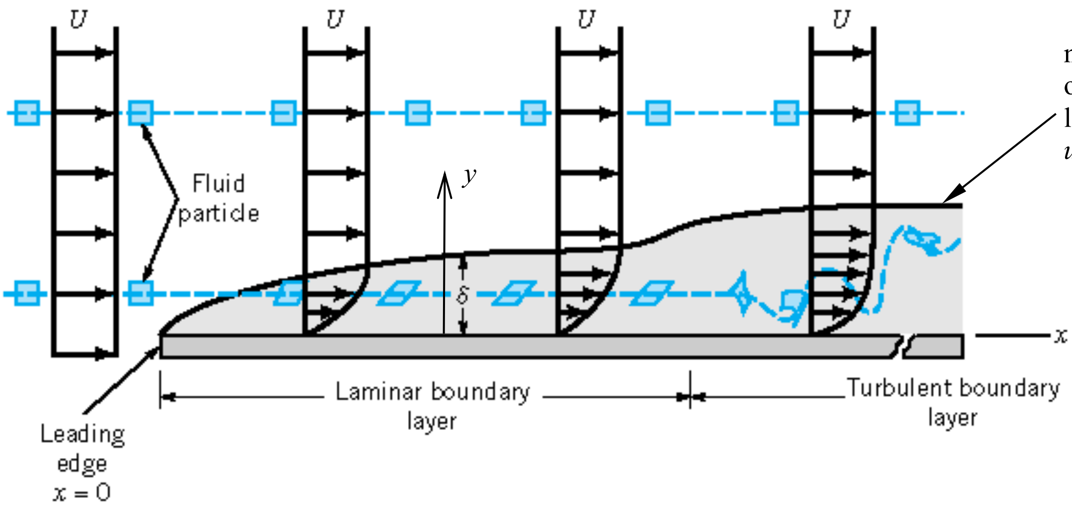
Flow of a uniform stream parallel to a flat plate. The larger the Reynolds number, the thinner the boundary layer along the plate at a given x-location.



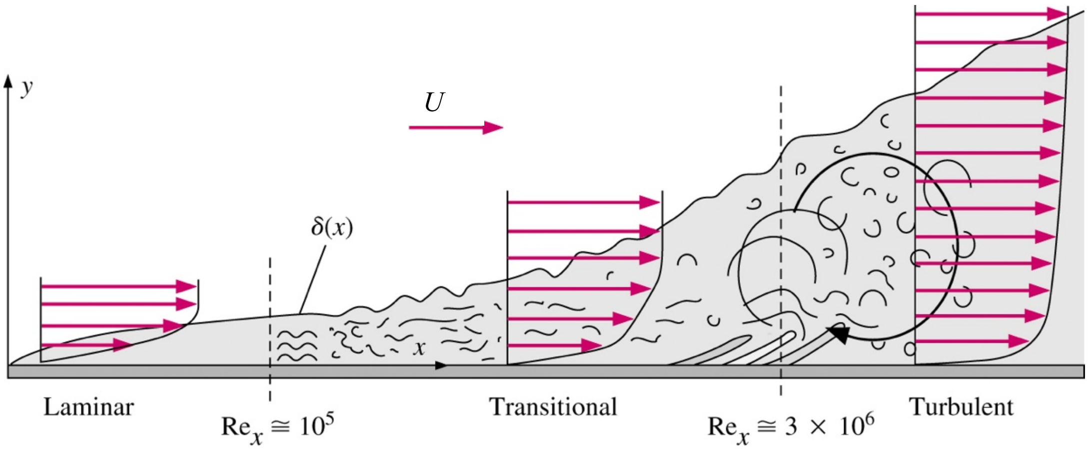
Flow past a circular cylinder; the boundary layer separates from the surface of the body in the wake for large Reynolds number.

B. Description of the Boundary Layer)

(1) Development of the Boundary Layer



nominal limit of boundary layer $u = 0.99\ U$



- On-coming flow is irrotational and has a uniform velocity *U*.

- The boundary layer starts out as a laminar boundary layer, in which fluid particles move in smooth layers and the velocity distribution is approximately parabolic. As the flow moves on, <u>the continual action of shear stress tends to slow down additional fluid particles, causing the boundary layer thickness to increase with distance downstream from the leading edge</u>. See below for a definition of the boundary layer thickness.

- The flow within the boundary layer is subject to wall shear, and dominated by viscous forces. The velocity gradient (hence the rotation of fluid particles) is the largest at the wall, and decreases with distance away from the wall, and tends to zero on matching with the main stream flow. Roughly speaking, the flow is said to be rotational within the boundary layer, but is irrotational outside the boundary layer.

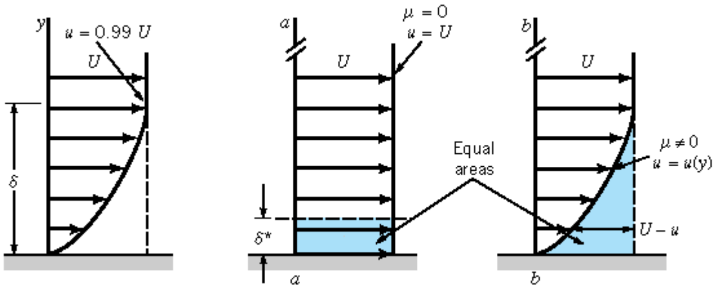
- As the thickness of laminar boundary layer increases, it becomes unstable and some eddying commences. These changes take place over a short length known as the transition zone.

- It finally transforms into a turbulent boundary layer, in which particles move in haphazard paths. Due to the turbulent mixing, the velocity distribution is much more uniform than that in the laminar boundary layer. The increase of thickness along the plate continues indefinitely but with a diminishing rate. If the plate is smooth (i.e., negligible roughness size), laminar flow persists in a very thin film called the viscous sub-layer in immediate contact with the plate and it is in this sub-layer that the greater part of the velocity change occurs.

(2) Thicknesses of the Boundary Layer

i) Boundary Layer Thickness δ

[Figure (a)] The velocity within the boundary layer increases to the velocity of the main stream asymptotically. It is conventional to define the boundary layer thickness δ as the distance from the boundary at which the velocity is 99% of the main stream velocity.



(a) (b)

There are other 'thicknesses', precisely defined by mathematical expressions, which are measures of the effect of the boundary layer on the flow.

ii) Displacement Thickness δ^*

It is defined by

[Figure (b)] δ^* is the distance by which the boundary surface would have to be shifted outward if the fluid were frictionless and carried at the same mass flowrate as the actual viscous flow.

Shown in figure (b) are two velocity profiles for flow past a flat plate – one if there were no viscosity (a uniform profile) and the other if there are viscosity and no-slip at the wall (the boundary layer profile). Because of the velocity deficit, $U - u$, within the boundary layer, the flowrate across section b-b is less than that across section a-a. If we displace the plate at section a-a by an appropriate amount δ^*, the flowrates across each section will be identical, which defines δ^* as shown above.

Conceptually one may 'add' this displacement thickness to the actual wall and treat the flow over the 'thickened' body as an inviscid flow. Let's look at book example 9.3.

iii) Momentum Thickness θ

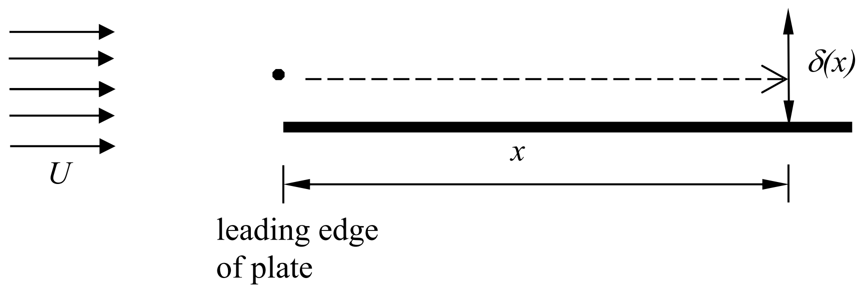
It is defined by

θ is the thickness of a layer of the main stream whose flux of momentum equals the deficiency in the boundary layer, equivalent to the loss of momentum flux per unit width divided by ρU^2 due to the presence of the growing boundary layer. The momentum thickness is often used when determining the drag on an object.

Note that when evaluating the above integrals for δ^* and θ, the upper integration limit can practically be replaced by δ.

C. Laminar Boundary Layer Over a Flat Plate

- Heuristic Analysis



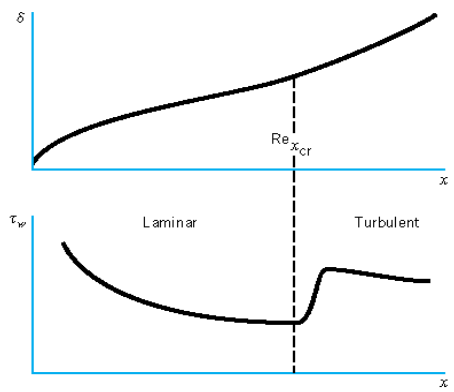
Consider steady flow past a flat plate at zero incidence. The effect of viscosity is to diffuse momentum normal to the plate. Consider a fluid element that is close enough to the wall to be influenced by viscosity. In travelling a distance x, it has been influenced by viscosity for a time $t \sim x/U$. The influence of viscosity will have spread laterally to a distance

The above analysis is rather crude, and does not yield a full equation for the growth of the boundary layer thickness. It however correctly describes one important relationship for the laminar boundary layer: $\delta / x \propto \left(\mathrm{Re}_x\right)^{-1/2}$ where $\mathrm{Re}_x \equiv Ux/\nu$ is the local Reynolds number in terms of the distance from the leading edge x. This relationship is found to be valid at a distance far behind the leading edge: $\delta / x \ll 1$.

The heuristic analysis can be further carried on to find relations for the wall stress:

The wall shear stress τ_w decreases with increase of x until the boundary layer turns turbulent. The local friction coefficient, which is defined as follows, is given by

While the numerical factor of 2 is far from the true value, the functional dependence of C_f on Re_x is correctly predicted.



- Exact Solution by Blasius (Section 9.2.2)
  A more rigorous analysis, using the technique of **similarity solution**, was developed by Blasius for the laminar boundary layer over a flat plate. While the details of the analysis are beyond the scope of this course, it is important to note the following results derived from Blasius' solution.

boundary layer thickness $\quad \dfrac{\delta(x)}{x} = \dfrac{5}{\left(\mathrm{Re}_x\right)^{1/2}}$

friction coefficient $\quad C_f \equiv \dfrac{\tau_w}{\frac{1}{2}\rho U^2} = \dfrac{0.664}{\left(\mathrm{Re}_x\right)^{1/2}}$

drag coefficient $\quad C_D \equiv \dfrac{\mathcal{D}}{\frac{1}{2}\rho U^2 L} = \dfrac{1.328}{\left(\mathrm{Re}_L\right)^{1/2}}$

where $\mathcal{D}$ is the skin friction drag force on unit width of a plate of length L:
$\mathcal{D} = \int_0^L \tau_w dx$.

D. The Boundary Layer Momentum-Integral Equation

By virtue of the property that the boundary layer thickness δ is much smaller than the streamwise length scale (say, L): $\delta / L \ll 1$, one may simplify the Navier-Stokes equations to obtain the boundary-layer approximation:

$$\begin{cases} \text{continuity} & \dfrac{\partial u}{\partial x} + \dfrac{\partial v}{\partial y} = 0 \\ x\text{-momentum} & u\dfrac{\partial u}{\partial x} + v\dfrac{\partial u}{\partial y} = -\dfrac{1}{\rho}\dfrac{\partial p}{\partial x} + \nu\dfrac{\partial^2 u}{\partial y^2} \\ y\text{-momentum} & 0 = -\dfrac{1}{\rho}\dfrac{\partial p}{\partial y} \end{cases}$$

with the boundary conditions:

$(u,v) = (0,0) \quad$ at $\quad y = 0$

$u = U,\ \ p = P \quad$ as $\quad y \to \infty$

(where U, P are the velocity and pressure of the inviscid flow just outside the boundary layer)

From the y-momentum equation, it is clear that <u>the pressure in the boundary layer is constant laterally across the layer</u> and equal to the near-wall pressure of the inviscid flow outside the boundary layer.

On integrating the x-momentum equation with respect to y from $y = 0$ to $y = \delta$, and after some algebra including the use of the continuity equation, one may obtain the Karman momentum integral equation

$$\boxed{\tau_w = \rho \frac{d}{dx}\left(U^2\theta\right) + \rho U \delta^* \frac{dU}{dx}}$$

where $\tau_w =$ wall shear stress

$\rho =$ density

$U =$ near-wall velocity of the outer inviscid flow

$\theta =$ momentum thickness $= \int_0^\delta \frac{u}{U}\left(1 - \frac{u}{U}\right) dy$

$\delta^* =$ displacement thickness $= \int_0^\delta \left(1 - \frac{u}{U}\right) dy$

$\delta =$ boundary layer thickness

This momentum integral equation is applicable to laminar, transitional or turbulent boundary layer.

In particular, in the absence of pressure gradient (e.g., flow over a flat plate), the free stream velocity $U =$ constant and $dU/dx = 0$, and therefore the momentum integral equation reduces to

by which the skin friction drag and drag coefficient are simply given by

$$\mathcal{D} = \int_0^L \tau_w dx = \rho U^2 \int_0^L \frac{d\theta}{dx} dx = \rho U^2 \theta_L, \qquad C_D = \frac{\mathcal{D}}{\frac{1}{2}\rho U^2 L} = \frac{\rho U^2 \theta_L}{\frac{1}{2}\rho U^2 L} = 2\frac{\theta_L}{L}$$

where θ_L is the momentum thickness at $x = L$.

(1) <u>Laminar Boundary Layer Over a Flat Plate Revisited – approximate solution by momentum integral equation</u>

It is remarkable that approximate solutions, which are reasonably close to the exact ones, can be obtained for the boundary layer thickness and drag coefficients from the momentum integral equation on adopting an assumed velocity profile

$$\frac{u}{U} = f(\eta)$$

where $\eta = \frac{y}{\delta(x)}$ is the y-coordinate normalized with respect to the local boundary layer thickness.

The steps are as follows:-

a) Find the relation between θ and δ

b) Find the wall shear stress from Newton's law of viscosity

c) Substitute θ and τ_w into the momentum integral equation

Integrating the above equation with respect to x, assuming that $\delta = 0$ at $x = 0$:

Furthermore,

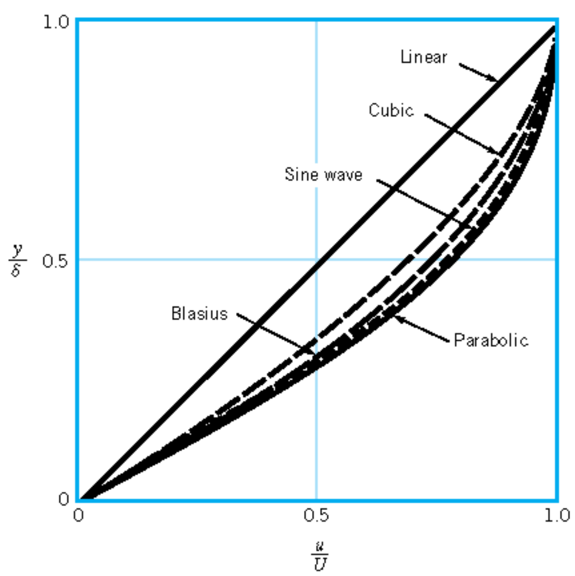
It turns out that the values of a and b are rather insensitive to the choice of the approximate velocity profile $u/U = f(\eta)$ as long as it is a reasonable one satisfying the boundary conditions.

Some assumed velocity profiles are

$$f(\eta) = \begin{cases} 2\eta - \eta^2 & \text{parabolic} \\ \dfrac{3}{2}\eta - \dfrac{\eta^3}{2} & \text{cubic} \\ \sin\left(\dfrac{\pi\eta}{2}\right) & \text{sine} \end{cases}$$

which satisfy

$$\begin{aligned} &f(0) = 0, \quad (\text{no-slip at } y = 0) \\ &f(1) = 1, \quad (u = U \text{ at } y = \delta) \quad . \\ &f'(1) = 0 \quad (\text{no stress at } y = \delta) \end{aligned}$$



■ **TABLE 9.2**

Flat Plate Momentum-Integral Results for Various Assumed Laminar Flow Velocity Profiles

Profile Character	$\delta \mathrm{Re}_x^{1/2}/x$	$c_f \mathrm{Re}_x^{1/2}$	$C_{Df} \mathrm{Re}_\ell^{1/2}$
a. Blasius solution	5.00	0.664	1.328
b. Linear $u/U = y/\delta$	3.46	0.578	1.156
c. Parabolic $u/U = 2y/\delta - (y/\delta)^2$	5.48	0.730	1.460
d. Cubic $u/U = 3(y/\delta)/2 - (y/\delta)^3/2$	4.64	0.646	1.292
e. Sine wave $u/U = \sin[\pi(y/\delta)/2]$	4.79	0.655	1.310

(2) Turbulent Boundary Layer Over a Flat Plate

The one-seven-power law, suggested by Prandtl, is used for the velocity profile in the turbulent boundary layer with zero pressure gradient:

$$\frac{u}{U} = \left(\frac{y}{\delta}\right)^{1/7} \qquad \text{or} \qquad f(\eta) = \eta^{1/7}$$

by which the momentum thickness is

$$\theta = \delta \int_0^1 f(1-f)\,d\eta = \delta \int_0^1 \eta^{1/7}\left(1-\eta^{1/7}\right)d\eta = \frac{7}{72}\delta$$

The one-seven-power law fails to describe the velocity profile at $y = 0$, where $\partial u / \partial y \to \infty$. The following empirical formula obtained for pipe flow can be adopted here:

$$\tau_w = 0.0225 \rho U^2 \left(\frac{\nu}{U\delta}\right)^{1/4}$$

Substituting θ and τ_w into the momentum integral equation, and integrating with respect to x:

$$\frac{4}{5}\delta^{5/4} = \left(\frac{72 \times 0.0225}{7}\right)\left(\frac{\nu}{U}\right)^{1/4} x + \text{constant}$$

It is assumed that the turbulent boundary layer starts from $x = 0$. (This is a contradiction to the fact that the boundary layer starts out as a laminar one, but this assumption has given good results.) Therefore, the constant = 0. Further simplification yields

$$\frac{\delta}{x} = \frac{0.370}{\left(\mathrm{Re}_x\right)^{1/5}}, \qquad \frac{\tau_w}{\rho U^2} = \frac{0.0288}{\left(\mathrm{Re}_x\right)^{1/5}}, \qquad C_D = \frac{0.072}{\left(\mathrm{Re}_L\right)^{1/5}}.$$

These results are valid for smooth flat plates with $5 \times 10^5 < \mathrm{Re}_L < 10^7$.

Note that for the turbulent boundary layer flow the boundary layer thickness increases with x as $\delta \sim x^{4/5}$ and the shear stress decreases as $\tau_w \sim x^{-1/5}$. For laminar flow these dependencies are $x^{1/2}$ and $x^{-1/2}$, respectively.

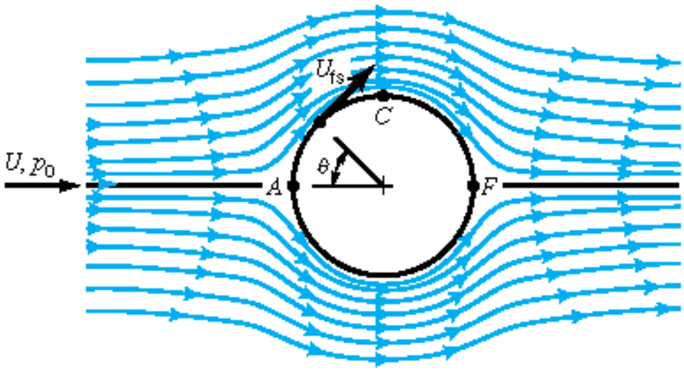
E. Effect of Pressure Gradient

The pressure in the streamwise direction (i.e., along the body surface) will not be constant if the body is not a flat plate. Consequently, the free stream velocity at the edge of the boundary layer U is also not a constant but a function of x. Whether the free-stream flow is accelerating or decelerating along the body surface will have dramatically different effects on the development of the boundary layer.

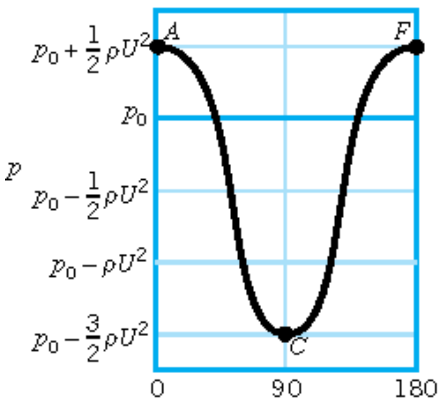
Let us re-examine flow past a circular cylinder, and find out what causes d'Alembert's paradox.

You may recall that inviscid flow past a circular cylinder has a symmetrical pressure distribution around the surface of the cylinder about the vertical axis. This results in a zero pressure drag, which is however not true in reality for any fluid with a finite viscosity. Such discrepancy is now referred to as d'Alembert's paradox.

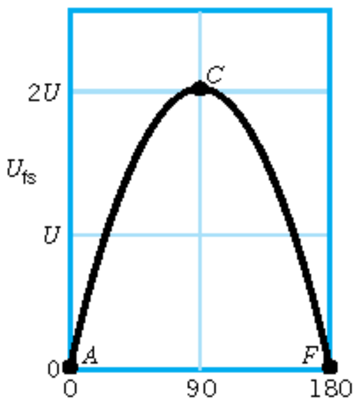
Despite the discrepancy, the potential theory helps to reveal that the pressure and hence the free-stream velocity U_{fs} on the cylinder's surface are not constant. From A to C, the pressure gradient is negative and the flow is accelerating, and from C to F, the opposite is true.



(a)

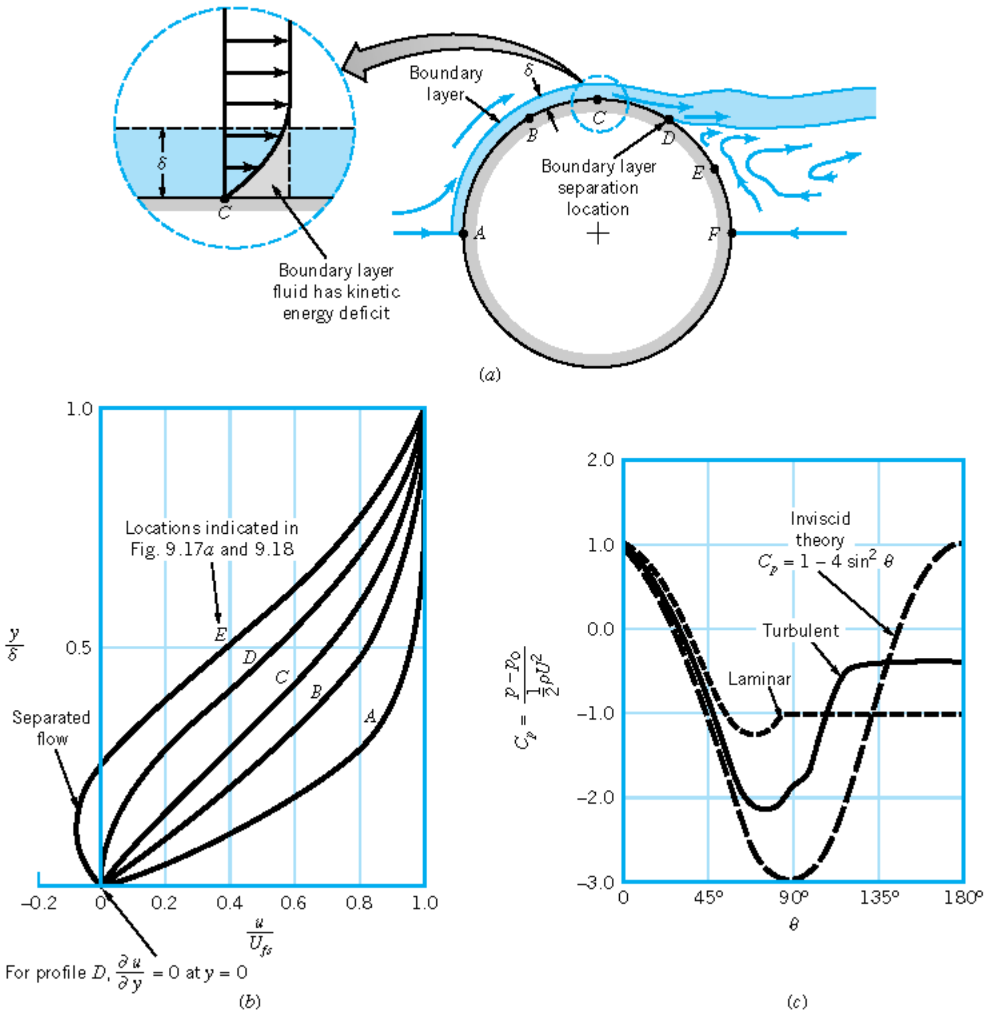


(b)



(c)

The real fluid flow past a circular cylinder is like this:



<u>Flow Past *A-B-C*</u>

- the streamlines are converging, i.e., flow is accelerating, and the free-stream velocity U reaches a maximum at C.
- the pressure is decreasing along the cylinder surface, i.e., $\partial p / \partial x < 0$, net pressure force is in forward direction, and the pressure gradient is said to be '**favorable**'.
- the accelerating flow tends to offset the 'slowing down' effect of the boundary on the fluid. Therefore, the rate of boundary layer thickening decreases and flow remains stable.

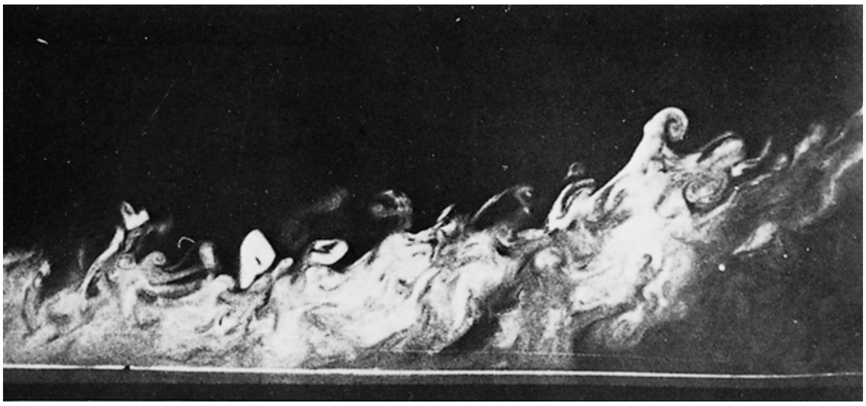
<u>Flow Past *C-D*</u>

- the streamlines are diverging, and the flow is retarding.
- the pressure is increasing along the cylinder surface, i.e., $\partial p / \partial x > 0$, net pressure force opposes the flow, and the pressure gradient is said to be '**adverse**' or '**unfavorable**'.
- it reduces the energy and forward momentum of the fluid particles in proximity to the surface, causing the thickness to increase sharply and fluid near the surface be brought to a standstill ($\partial u / \partial y$ at the surface is zero) at D. See figure (b).

Flow Past *D-E-F*

- flow close to the cylinder surface starts to reverse at *D* (separate point), i.e., fluid no longer to follow the contour of the surface. The phenomenon is termed **separation**.
- large irregular eddies formed in the reverse flow (**the wake**), in which much energy is lost to heat.
- the pressure in the wake remains approximately the same as at the separation point *D*, and is therefore lower than that predicted by the inviscid theory (see figure c). This lowering of pressure behind the cylinder resulting from flow separation leads to a net pressure drag on the cylinder. This explains d'Alembert's paradox. Note that the wider the wake, the larger the pressure drag, and vice versa.

(a)

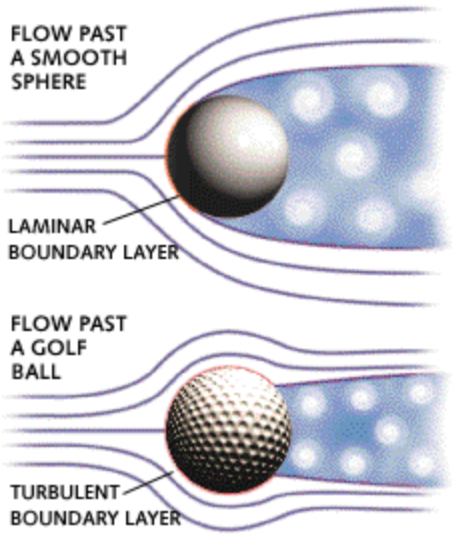


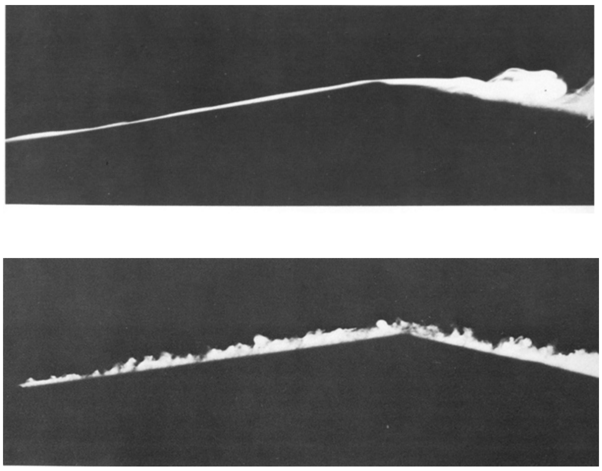
(b)

Influence of a strong pressure gradient on a turbulent flow: (a) flow is relaminarized by a negative (favorable) pressure gradient; (b) the boundary layer is thickened by a positive (unfavorable) pressure gradient.

<u>Further remarks about flow separation</u>

- separation can occur only under an adverse pressure gradient and when the fluid is viscous.
- separation occurs with both laminar and turbulent boundary layers. Laminar boundary layer is more prone to separation than turbulent boundary layer. Thus, as shown in figure (c) on page 11, the turbulent boundary layer can flow farther around the cylinder before it separates than the laminar boundary layer. Therefore the wake size will be narrower if the flow is turbulent at the separation point than if it is laminar. This explains why it is desirable to have dimples on a golf ball, which can effectively reduce the drag by inducing a narrower turbulent wake behind the ball.





Turbulent boundary layers are more resistant to flow separation than are laminar boundary layers exposed to the same adverse pressure gradient. The laminar boundary layer (upper) cannot negotiate the sharp turn of 20^{o}, and separates at the corner (flow is from left to right). The turbulent boundary layer (lower) on the other hand manages to remain attached around the sharp corner.

F. Drag

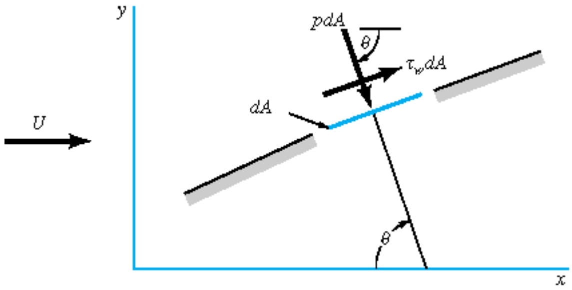
Any object moving through a fluid (or a stationary object immersed in a viscous flow) will experience a drag, $\mathcal{D}$ – a net force in the direction of flow due to the pressure and shear forces on the surface of the object.

Drag = Pressure Drag + Skin Friction Drag

where

Pressure Drag = resultant force arising from the non-uniform and asymmetrical pressure distribution around the surface the body. It is also called form drag as it depends on the form or the shape of the body.

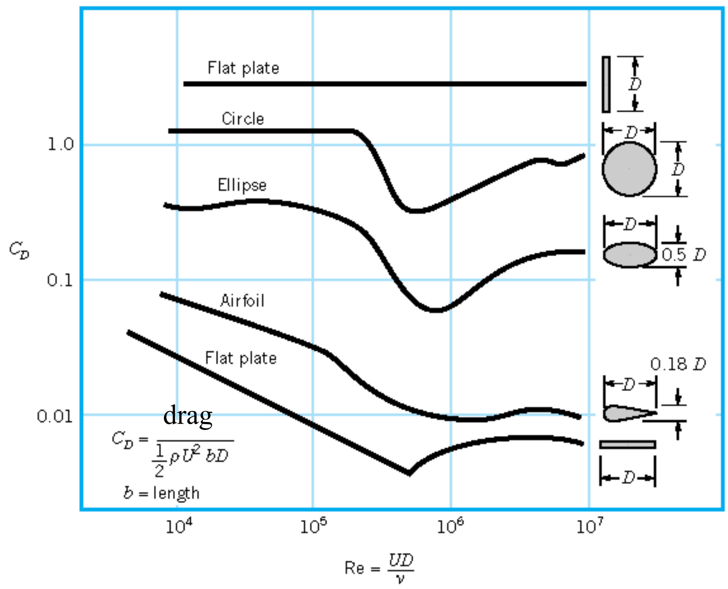
Skin Friction Drag = resultant force due to fluid shear stress on the surface of the object.



The drag coefficient C_D is given by the ratio of the total drag force to the dynamic force

$$C_D = \frac{\mathcal{D}}{\frac{1}{2}\rho U^2 A}$$

where U = relative velocity of fluid far upstream of the object,
A = <u>frontal area</u> – the projected area of the object when viewed from a direction parallel to the oncoming flow if it is a blunt (or bluff) object (e.g., a cylinder); or the <u>planform area</u> – the projected area of the object when viewed from above it if it is a streamlined object (e.g., a flat plate).



Typically the drag coefficient depends on

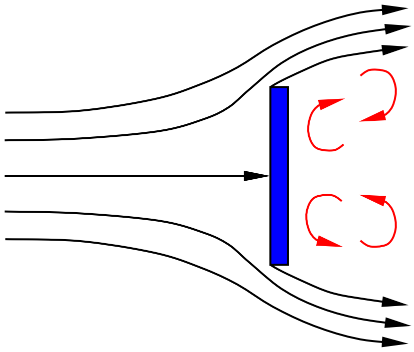
(i) the shape of the object,

(ii) orientation of the object with the flow (e.g., a flat plate normal to flow has a different C_D than a flat plate parallel to flow),

(iii) the Reynolds number $\mathrm{Re} = UD/\nu$ where D is a characteristic dimension of the object,

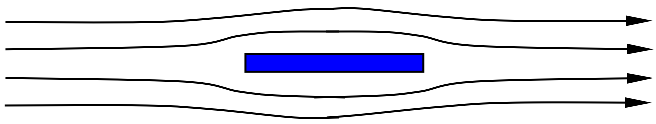
(iv) surface roughness if the drag is dominated by skin friction and the boundary layer is turbulent.

<u>Flow Past a Flat Plate</u>

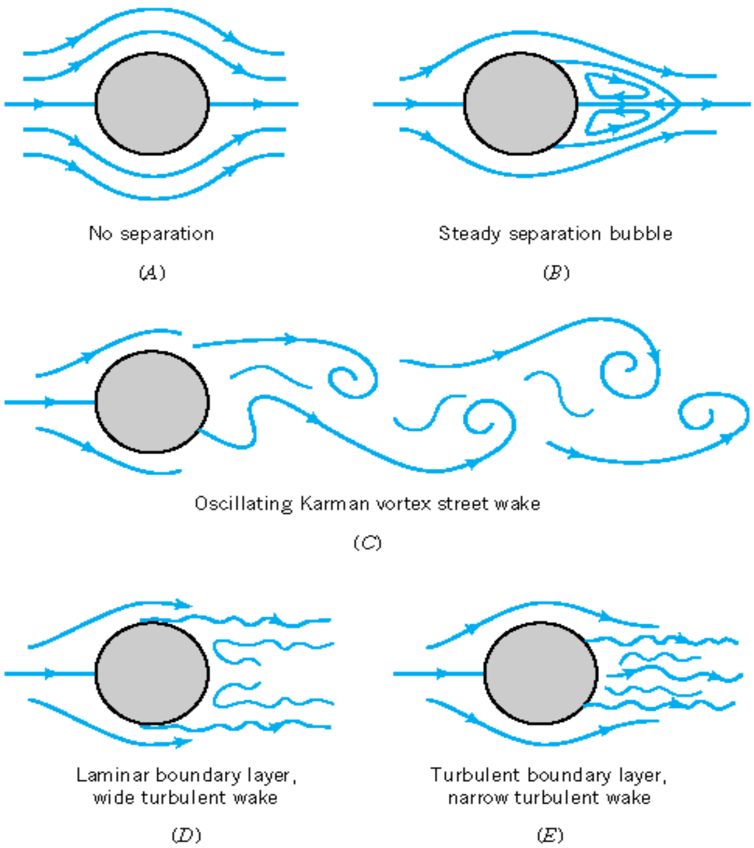


When a flat plate is held normal to flow, the flow is separated upon past over the plate. A region of eddying motion (wake) is formed at the rear of the plate, the pressure there being much reduced. Therefore the pressure drag is dominant, and the plate is a bluff body in this position. The drag shows little dependence on the Reynolds number.

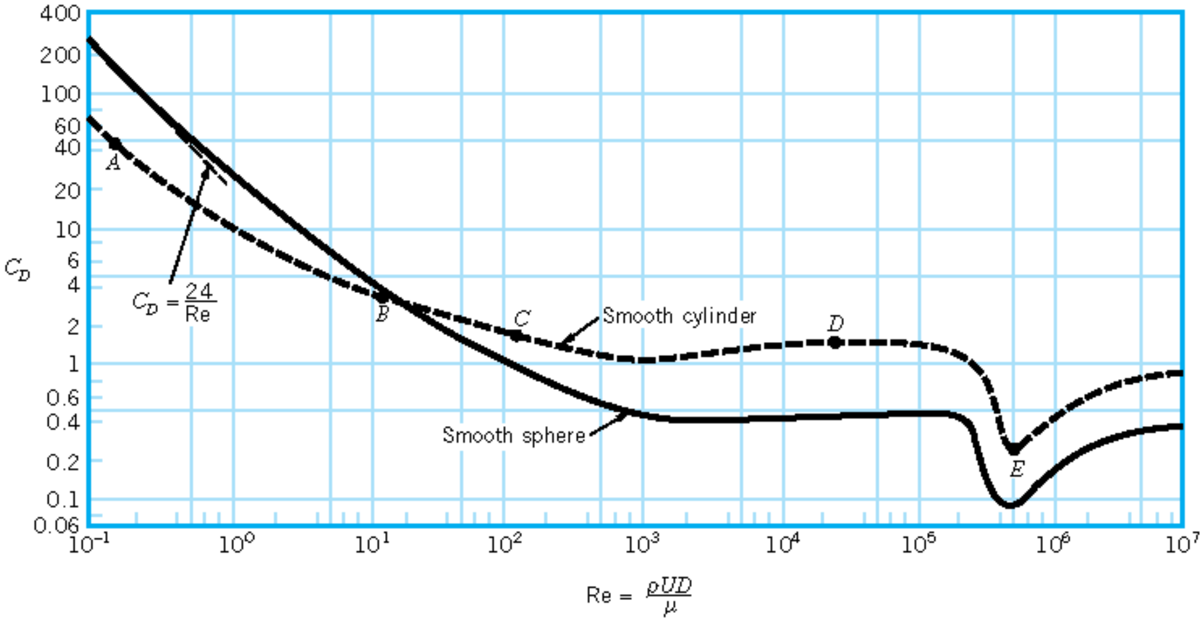
When a flat plate is held parallel to flow, formation of the boundary layer over the plate is appreciable and flow separation is negligible. Therefore the skin friction drag is significant. The plate is a streamlined body in this position. The drag coefficient increases when the boundary layer becomes turbulent.



<u>Flow Past a Circular Cylinder/Sphere</u>



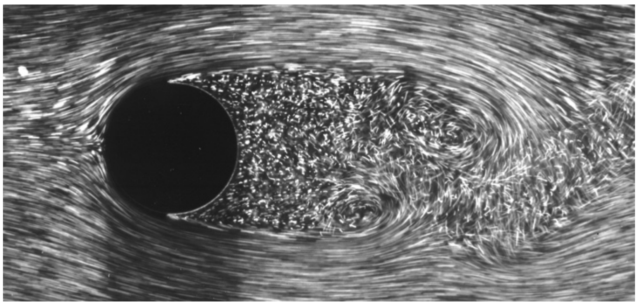
(b)



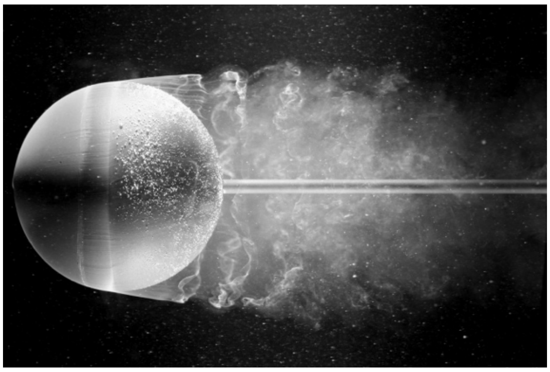
(*a*)

- $Re \le 1$
  - creeping flow
  - no flow separation
  - C_D decreases with increasing Re ($C_D = 24/Re$ for a sphere)

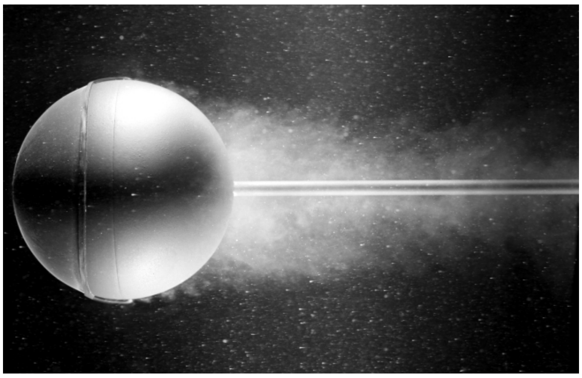
  (Note that a decrease in the drag coefficient with Re does not necessarily imply a corresponding decrease in drag. The drag force is proportional to the square of the velocity, and the increase in velocity at higher Re will usually more than offset the decrease in the drag coefficient.)
- Re = 10
  - separation starts occurring on the rear of the body forming a pair of vortex bubbles there
  - vortex shedding begins at Re $\cong$ 90, leading to an oscillating Karman vortex street wake (see next page)
  - region of separation increases with increasing Re
  - C_D continues to decrease with increasing Re until Re = 10^3, at which pressure drag dominates
- $10^3 < Re < 10^5$
  - C_D remains relatively constant, which is a characteristic behavior of blunt bodies
  - flow in the boundary layer is laminar, but the flow in the separated region is highly turbulent, thereby a wide turbulent wake
- $10^5 < Re < 10^6$
  - a sudden drop in C_D somewhere within this range of Re
  - this large reduction in C_D is due to the flow in the boundary layer becoming turbulent, which moves the separation point further on the rear of the body, reducing the size of the wake and hence the magnitude of the pressure drag. This is in sharp contrast to streamlined bodies, which experience an increase in the drag coefficient (mostly due to skin friction drag) when the boundary layer turns turbulent.



Laminar boundary layer separation with a turbulent wake for flow past a circular cylinder at Re = 2000.



(a)



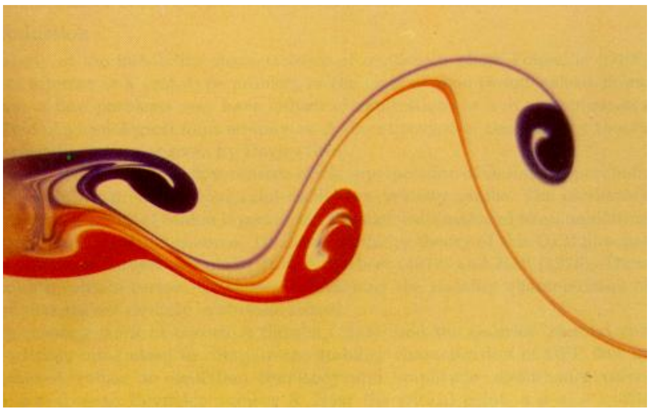
(b)

Flow over (a) a smooth sphere at Re = 15,000, and (b) a sphere at Re = 30,000 with a trip wire; the delay of boundary layer separation is clearly seen by comparing these two photographs. The delay of separation in turbulent flow is caused by the rapid fluctuations of the fluid in the transverse direction, which enables the turbulent boundary layer to travel farther along the surface before separation occurs, resulting in a narrower wake and a smaller pressure drag. Recall also that turbulent flow has a fuller velocity profile as compared to the laminar case, and thus it requires a stronger adverse pressure gradient to overcome the additional momentum close to the wall.

Karman Vortex Streets

The Karman vortex street is one of the best-known vortex patterns in fluid mechanics. The vortex street is just a special type of unsteady separation over bluff bodies such as a cylinder. The vortex street is highly periodic having a frequency which is proportional to *U/D*, where *D* is the length of the bluff body measured transverse to the flow and *U* is the incoming flow speed. This periodicity is responsible for the "singing" of telephone wires. In fact, vortex streets are almost always involved when the wind generates a fairly pure tone as it blows over obstacles.

A practical consequence of the regular, periodic flow is that the forces on the body are also periodic. Because the flow is asymmetric fore and aft as well as in the direction transverse to the flow, the body will experience both an oscillating drag and lift. If the frequency of the shedding is close to a structural frequency, resonance can occur, usually with unpleasant results.



Flows in Porous Media

1 Introduction

Porous media are solid materials with internal pore structures. The pores can be either empty or filled with fluids. Porous structures vary significantly among different media (Figure 1). A structure with a regular array of cylindrical pores can be found in micro- or nanofabricated materials. A foam structure is composed of a continuous solid phase with interconnected channels or isolated pores and is often observed as a sponge. A granular structure, exhibited by a pile of sand, consists of solid particles and the void space between them. A fiber matrix is the primary structure in polymeric gels. Biological tissues can contain several of these structures simultaneously.

There are three compartments in biological tissues: blood and lymph vessels, cells, and interstitium (Figure 2). The interstitial space can be further divided into the extracellular matrix and the interstitial fluid. Although the volume fraction of each compartment is tissue-dependent, it is generally less than 10% for the vascular space, which is smaller than those for the other two compartments. The extravascular region (cells and the interstitium) can be considered a porous medium, with pores saturated with interstitial fluid.

2 Porosity, tortuosity, and available volume fraction

Porous media can be characterized by their specific surface and porosity, respectively defined as

$$s = \frac{\text{Total interface area}}{\text{Total volume}} \tag{1}$$

and

$$\epsilon = \frac{\text{Void volume}}{\text{Total volume}}. \tag{2}$$

Note that the unit of s is one over length and that ϵ is dimensionless. The void volume is the total volume of the void space in a porous medium; the interface is the border between solid and void spaces. Both s and ϵ depend on the structure of pores.

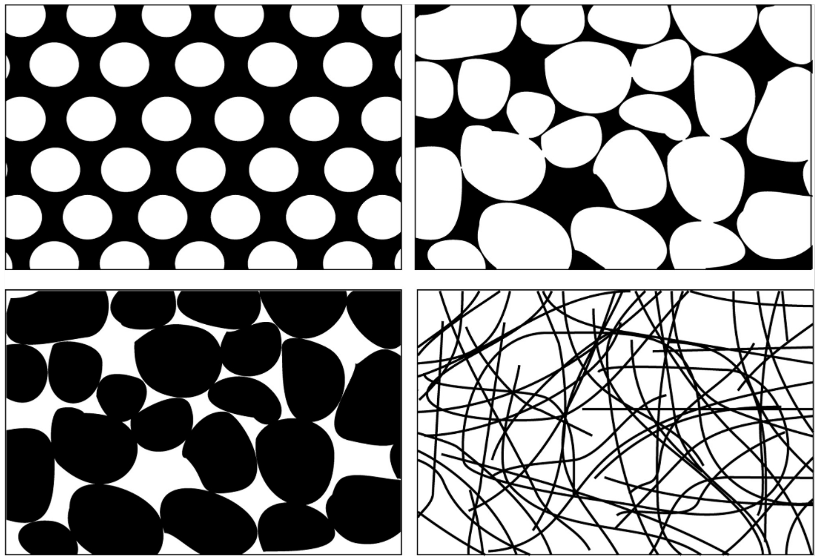


FIG. 1: Examples of porous structures. Upper left: a regular array of cylindrical pores; upper right: a foam structure; lower left: a granular structure; and lower right: a fiber matrix. In all examples, the white regions represent void spaces or the fluid phases of the media, and the black regions represent the solid phases.

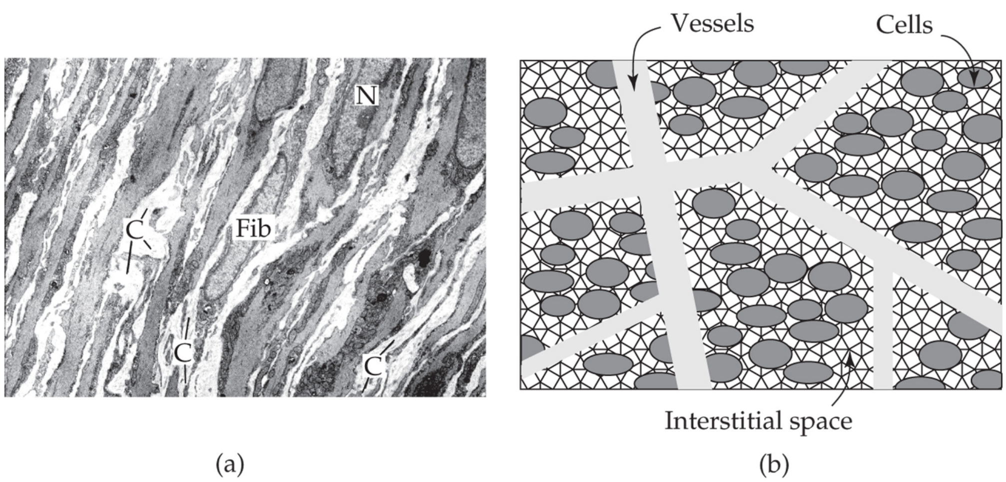


FIG. 2: Compartments in biological tissues. (a) An electron micrograph of smooth muscle tissues, where Fib indicates fibroblast, N indicates the nucleus of smooth muscle cells, and C indicates collagen fibrils. Blood vessels are not shown in this figure. (b) A schematic of biological tissues. The vessels can be either blood or lymph vessels. The cells include all populations in the tissue.

Some porous media, such as biological tissues, are deformable under mechanical loads. Material deformation can change the spatial distribution of the porosity. Thus, the porosity may vary both spatially and temporally. If a porous material is homogenous, its porosity can be easily calculated. We demonstrate this in Example 1 below.

Example 1: Determine the specific surface and the porosity of a porous medium with uniformly distributed cylindrical pores. Assume that the pores are parallel to each other (Figure 3), the diameter of pores is d, L is the length of the pores, and the number of cylinders per unit cross-sectional area is n_A. N is the total number of cylinders in the material and is equal to the product of the cross-sectional area A and n_A.

Solution: The void volume is the space within the cylinders. To calculate the specific surface and the porosity, we must first determine the total interface area and the void volume. We have

(3)

(4)

The specific surface and the porosity can be calculated by substituting Equations (3) and (4) into Equations (1) and (2), respectively. The results are

(5)

and

(6)

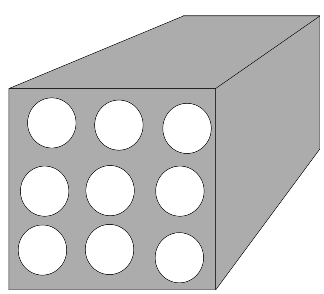


FIG. 3: A porous medium with cylindrical pores.

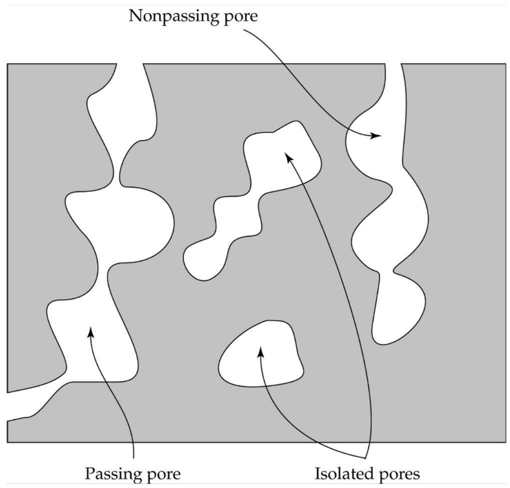


FIG. 4: Classification of pores based on their connections. The passing pores connect to two boundaries of the rectangular material. A nonpassing pore connects to only one subdomain of the outer surface. Passing and nonpassing pores together are called penetrable pores. Isolated pores have no connections to the outer surface of the porous media.

The porosity is a measure of the average void volume fraction in a specific region of porous medium. It does not provide any information on how different pores are connected or on how many pores are available for water and solute transport. Therefore, we divide the pores into the following three categories (Figure 4):

$$\begin{aligned} &\text{penetrable pores} : \text{passing pores}, \text{nonpassing pores} \\ &\text{isolated pores.} \end{aligned} \tag{7}$$

Based on this classification of pores, the porosity can be expressed as the sum,

$$\epsilon = \epsilon_i + \epsilon_p + \epsilon_n, \tag{8}$$

where the subscripts i, p, and n indicate isolated, passing, and nonpassing pores, respectively. Isolated pores are not accessible to external solvents and solutes; therefore, they can sometimes be considered as part of the solid phase in transport analysis. In this case, the void volume is defined as the total volume of penetrable pores.

The path length between points A and B in a porous medium is measured by the distance between these points through connected pores. The shortest path length, L_{min}, can be characterized by the geometric tortuosity

$$T = \left(\frac{L_{min}}{L}\right)^2, \tag{9}$$

where L is the straight-line distance between A and B. The tortuosity depends on the locations of A and B and on the structures of porous media. By definition, T is always greater than or equal to unity.

Not all penetrable pores are accessible to solutes. Such accessibility will depend upon the molecular properties of the solutes. For example, a pore will be inaccessible to a solute if the solute molecule is larger than the pore. The portion of accessible volume that can be occupied by the solute is called the available volume. For a solute, the ratio of the available volume to the total volume is defined as the available volume fraction:

$$K_{AV} = \frac{\text{Available volume}}{\text{Total volume}}. \tag{10}$$

By definition, K_{AV} is molecule-dependent and always smaller than the porosity ϵ. There are three possible scenarios that can cause K_{AV} to be less than ϵ. First, the centers of the solute molecules cannot reach the solid surface in the void space. In this case, the difference between the total void volume and the available volume can be estimated as the product of the area of the solid surface and the distance Δ between the solute and the surface (Figure 5). The distance Δ is equal to the radius of the solute if the solute can come into contact with the surface. However, many biological surfaces are electrically charged. Solutes that have the same charge as the solid surface does may not be able to reach the surface. In such cases, Δ is larger than the radius of the solute.

The second scenario that can cause K_{AV} to be less than ϵ is the situation in which some of the void space is smaller than the solute molecules. The third scenario is the inaccessibility of large penetrable pores surrounded by pores that are smaller than the solutes.

The ratio of the available volume to the void volume is defined as the partition coefficient Φ of the solutes. The partition coefficient is a measure of solute partitioning at equilibrium between external solutions and the void space in porous media. This parameter is different from that with the same name used in chemistry, in which the partition coefficient characterizes the partitioning

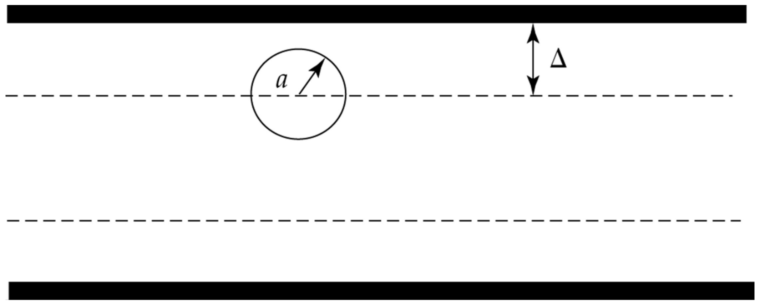


FIG. 5: Steric exclusion of solutes near the surface of solid phase in porous media. The circle represents the solute molecule. Δ is the distance between the solute and the surface.

of solutes between two liquid phases, such as oil and water. By definition, the partition coefficient in porous media is related to K_{AV} and ϵ via the formula

$$\Phi = \frac{\text{Available volume/Total volume}}{\text{Void volume/Total volume}} = \frac{K_{AV}}{\epsilon}. \tag{11}$$

Example 2: Determine the partition coefficient and the available volume fraction of a spherical solute in the same porous medium as that in Example 1. Assume that the diameter of the solute is b and that electric charge-charge interactions are negligible.

Solution: The void volume in each cylinder is equal to $L\pi d^2/4$, and the available volume is equal to $L\pi(d/2 - b/2)^2$. Thus, the partition coefficient can be calculated as

(12)

where $\lambda = b/d$ is the ratio of radius of the solute to that of the pore. The available volume fraction is equal to the product of the partition coefficient and the porosity, where the porosity was determined in Example 1. Thus,

(13)

Note that, if $\lambda \ll 1$, $\Phi \approx 1$, and $K_{AV} \approx \epsilon$.

---

3 Fluid flow in porous media

3.1 Darcy's law

Fluid flow in porous media has been studied for more than a century. The interaction between solid and liquid phases in porous media was first quantified by Darcy in 1856, nearly at the same time that Fick developed his theory of molecular diffusion. In a study of water percolating through sand, Darcy discovered that the flow rate was proportional to the pressure gradient. This empirical relationship, called Darcy's law, is found to be valid in many porous media and was theoretically derived by other investigators later, on the basis of mechanical analyses of fluid flow in porous media. The derivation shows that Darcy's law is invalid for non-Newtonian fluids, for Newtonian liquids at high velocity, and for gases at very low and very high velocities. The derivation also reveals that Darcy's law neglects the friction within the fluid and the exchange of momentum between the fluid and solid phases. Except for the friction within the fluid, which is discussed in Section 3.2, these exceptional cases of Darcy's law have rarely been observed in the interstitium of biological tissues. Therefore, Darcy's law has been widely used in the analysis of interstitial fluid flow.

The movement of fluid molecules in porous media follows tortuous pathways in the void space (Figure 6). To describe the fluid flow in porous media, two approaches can be used. One is to numerically solve the governing equations for fluid flow in individual pores if the structures of pore networks are known. The other approach is to assume that a porous medium is a uniform material. In this so-called continuum approach proposed by Darcy in 1856, there are three length scales. The first one is the average size δ of the pores. The second is the distance L over which macroscopic changes of physical quantities (e.g., fluid velocity and pressure) must be considered. In most cases, L is chosen to be the characteristic linear dimension of the porous medium (e.g., the size of tissues or the distance between adjacent blood vessels). The continuum approach requires that L be at least two orders of magnitude larger than δ so that there may exist a third length scale, l, between δ and L. To define l, we consider a volume V_i of dimension l in the porous medium. The volume fraction of the void space is then the volumetric porosity ϵ, and the volume fraction of the solid phase is equal to $1 - \epsilon$. When l is close to δ, the porosity ϵ is highly sensitive to the value of

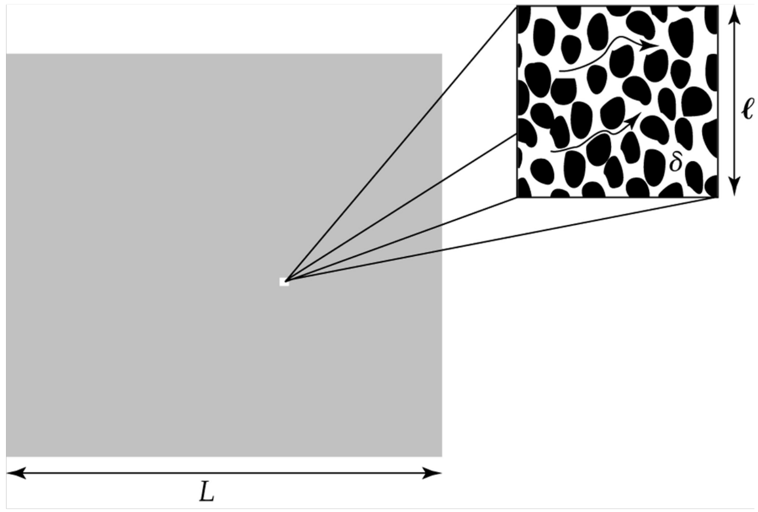


FIG. 6: A sketch of fluid and solute transport in a porous medium. The medium is shown as a filled gray area. Medium can have any structure but it is represented by a granular structure in this figure. The solid phase is shown as the black areas in the insert. Fluid and solutes can move between solid particles. The characteristic size of pores, δ, is equal to the average distance between adjacent particles. A small volume with a dimension, l, is shown as the insert. The characteristic distance of transport, L, is the size of the gray area. The curved arrows in the insert indicate examples of transport pathways.

l. When l is increased gradually, the fluctuation in ϵ will decrease. There exists a value l_0 of l beyond which ϵ is a smooth function of l, although it still fluctuates with a very small amplitude due to the random distribution of pore size in the volume V_i. The continuum approach requires that $\delta \ll l_0 \ll L$. In biological tissues, $\delta < 0.1\mu m$, $l_0 \sim 1\mu m$, and $L \sim 100\mu m$ to $10cm$. Thus, transport in biological tissues can be studied with the continuum approach.

The volume with dimension l_0 is called the representative elementary volume (REV). In porous media, the REV can be taken to be a point, called a material point, since $l_0 \ll L$ (Figure 6). In that case, the details of pore structures are neglected, and each spatial point simultaneously contains two phases: a void phase with a volume fraction of ϵ, and a solid phase with a volume fraction of $1 - \epsilon$. For the present discussion, we consider only porous media with pores filled with fluid. Thus, we do not specifically distinguish the void versus fluid phases. At each material point, any physical quantity can be defined as the volume average of the same quantity defined in a pure medium.

Fluid transport in porous media must satisfy the law of mass conservation. For a pure, incompressible fluid, the mass balance equation in the liquid phase states simply that the divergence of the fluid velocity is equal to zero. The same equation is also valid in a porous medium, if there is no fluid production (know as a source) or fluid consumption (known as a sink) in the medium. However, sources and sinks are often present in biological tissues. For example, fluid is exchanged between interstitial space and the blood or lymph vessels. Thus, the mass balance equation, $\nabla \cdot \boldsymbol{v} = 0$, needs to be modified by adding a source term and a sink term:

$$\nabla \cdot \boldsymbol{v} = \phi_B - \phi_L. \tag{14}$$

Here, $\boldsymbol{v}$ is the fluid velocity averaged in the REV. In Equation (14), ϕ_B and ϕ_L are rates of volumetric flow in sources and sinks, respectively, per unit volume of a porous medium. In biological tissues, they represent the rate of fluid flow per unit volume from blood vessels into the interstitial space and from the interstitial space into lymph vessels, respectively.

The equation for momentum balance in porous media is Darcy's law, introduced earlier. In a homogeneous and isotropic medium, Darcy's law can be written as

$$\boldsymbol{v} = -H\nabla p, \tag{15}$$

where ∇p is the gradient of the hydrostatic pressure and H is a constant defined as the hydraulic conductivity. Note that p is defined as the average quantity within the fluid phase in the REV. For nonisotropic and heterogeneous media, H is a tensor and it depends upon the location in the medium.

Substituting Equation (15) into (14), we get

$$-\nabla \cdot (H\nabla p) = \phi_B - \phi_L. \tag{16}$$

Equations (15) and (16) are the governing equations for fluid flow in rigid porous media. A special case of Equation (16) is

$$\nabla^2 p = 0, \tag{17}$$

when H is a constant and $\phi_B = \phi_L = 0$. In this case, the interstitial fluid pressure is governed by a Laplace equation.

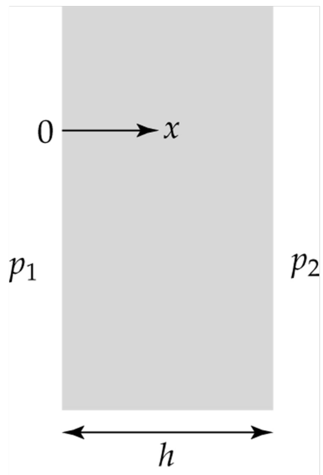


FIG. 7: One-dimensional flow through a porous medium.

Example 3: Consider one-dimensional flow through a porous medium with hydraulic conductivity H (Figure 7). The thickness of the medium is h. The pressures at $x = 0$ and $x = h$ are p_1 and p_2, respectively. Determine the pressure and velocity distributions in the medium. Assume $\phi_B = \phi_L = 0$.

Solution: Let us start from the mass balance and momentum equations. In this problem, $\phi_B = \phi_L = 0$ and the flow is one dimensional. Equations (15) and (16) then become, respectively,

(18)

and

(19)

Integrating Equation (19) twice, we get

(20)

where a_1, and a_2 are constants that can be determined from the boundary conditions of the pressure:

(21)

Thus,

(22)

The velocity profile can be obtained by substituting Equation (22) into Equation (18). The result is

(23)

For one-dimensional flow, one can see that the pressure decreases linearly as a function of distance and the velocity is a constant in the medium.

If the gravitational force is not negligible, then Darcy's law must be modified to read

$$\boldsymbol{v} = -H(\nabla p - \rho \boldsymbol{g}), \tag{24}$$

where ρ is the density of the fluid and $\boldsymbol{g}$ is the acceleration due to gravity. Equation (24) indicates that the gravitational force can be neglected only when $\Delta p/L \gg \rho g$, where L is the distance over which the change in pressure is Δp.

The hydraulic conductivity is inversely proportional to the viscosity of the fluid μ. The product of H and μ is defined as the specific hydraulic permeability k and depends only upon microscopic structures of the porous medium. For porous media with simple structures, H or k can be predicted theoretically.

Example 4: Determine the hydraulic conductivity and specific hydraulic permeability in the same porous medium as that discussed in Example 1. Assume that the liquid is a Newtonian fluid.

Solution: Fluid flow in a circular cylinder is governed by Poiseuille's law, which predicts the dependence of the flow rate q on the pressure gradient:

(25)

The total flow rate Q across the porous medium is equal to the sum of the flow rates across all of the cylinders; that is, $Q = An_A q$, where A is the cross-sectional area of the porous medium and n_A is the number of pores per unit area. By definition, the velocity v in the porous medium in the axial direction of cylinders is equal to the total flow rate per unit area. Thus,

(26)

Comparing this equation with Darcy's law (Equation (15)), we derive

$$\tag{27}$$

In Example 4, we assumed the cross section of pores to be circular. Analytical solutions of fluid flow in noncircular, cylindrical pores have been derived by Kozeny (1927), on the basis of the Navier-Stokes equation. Kozeny found that

$$H = \frac{c\epsilon^3}{\mu s^2}, \tag{28}$$

where ϵ is the porosity, s is the specific surface, defined by Equation (1), and c is a shape factor, also called the Kozeny constant. Some examples of c are given in Table 8.1.

For noncylindrical pores (Figure 1), a more general equation for calculating K is the Kozeny-Carman equation,

$$H = \frac{\epsilon^3}{G\mu s_0^2(1-\epsilon)^2}, \tag{29}$$

where G is also called the Kozeny constant in the literature. The variable s_0 is the Carman-specific surface, defined as the area of the surface that is exposed to the fluid per unit volume of the solid phase. For porous media with parallel, cylindrical pores, $s = s_0(1-\epsilon)$ and G is equal to $1/c$. In this case, Equation (29) reduces to Equation (28). In other porous media, G is equal to $1/(Tc)$ (which is less than $1/c$), where T is the tortuosity defined in Equation (9).

TABLE 8.1

Values of c for Different Shapes of Cylindrical Pores

Shape	c
Circle	0.5
Square	0.5619
Equilateral triangle	0.5974
Strip	0.6667

3.2 Brinkman equation

The interstitial space can be considered a network of channels filled with porous media. Fluid flow in such channels may not be modeled correctly by Darcy's law, because the fluid velocity in Darcy's model does not satisfy the no-slip boundary condition on the channel wall. From the physical point of view, Darcy's law assumes that the viscous resistance at the fluid-solid interface is much larger than that within the fluid. This assumption is valid, however, only when the specific permeability k of porous media is low. That condition in turn implies a high fiber concentration in a fiber matrix. The viscous stress within the fluid may not be negligible when k is large. In this case, the momentum equation must be derived again, using the Stokes equation for low-Reynolds-number flow. The result, called the Brinkman equation, is

$$\mu\nabla^2\boldsymbol{v} - \frac{1}{H}\boldsymbol{v} - \nabla p = 0. \tag{30}$$

Darcy's law, Equation (15), can be considered a special case of the Brinkman Equation (30) wherein the first term can be neglected. In this case, the specific permeability k of the porous medium is much less than the square of the characteristic length L over which macroscopic changes in fluid velocity must be considered.

Example 5: Assume that the interstitial space between two cells can be considered to be a porous channel bounded by two parallel plates (Figure 8). The effective hydraulic conductivity in the channel $H_{channel}$, defined as the ratio of the fluid flux to the pressure gradient, depends on the specific hydraulic permeability in the porous medium k and the interaction of fluid with the channel wall. The channel height h is much smaller than the size of cells. Therefore, the flow can be assumed to be unidirectional. Derive the velocity profile and the expression of $H_{channel}$ as a function of k, h, and the dynamic viscosity of the fluid μ.

Solution: The flow is unidirectional in the channel and is governed by the mass balance equation and the Brinkman equations. With $\phi_B = \phi_L = 0$, Equations (14) and (30) become, respectively,

$$\frac{\partial v_x}{\partial x} = 0, \tag{31}$$

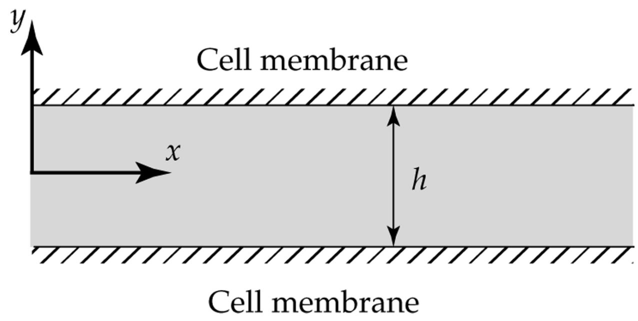


FIG. 8: Fluid flow in a channel that separates two cells. The channel represents the interstitial space filled with extracellular matrix and interstitial fluid. It can be modeled as a uniform porous medium. The height of the channel is denoted by h.

$$-\frac{\partial p}{\partial y} = 0, \tag{32}$$

$$\mu \frac{\partial^2 v_x}{\partial y^2} - \frac{\mu}{k} v_x - \frac{\partial p}{\partial x} = 0. \tag{33}$$

Equations (31) and (32) indicate that v_x is independent of x and p is independent of y, respectively. Thus, Equation (33) becomes

$$\mu \frac{d^2 v_x}{dy^2} - \frac{\mu}{k} v_x = \frac{dp}{dx}. \tag{34}$$

The right hand side of the equation is a function of x, whereas the left of the equation is function of y. Therefore, the only possibilities that both sides are independent of x and y (i.e., they are equal to constant). If we denote $B = dp/dx$ and rearrange terms in Equation (34), we get

$$\frac{d^2 v_x}{dy^2} - \frac{1}{k} v_x = \frac{B}{\mu}. \tag{35}$$

The general solution of Equation (35) is

$$v_x = c_1 \sinh\left(\frac{y}{\sqrt{k}}\right) + c_2 \cosh\left(\frac{y}{\sqrt{k}}\right) - \frac{k}{\mu} B, \tag{36}$$

where c_1 and c_2 are constants that can be determined from the boundary conditions of v_x, namely,

$$\frac{dv_x}{dy} = 0 \quad \text{at} \quad y = 0 \tag{37}$$

and

$$v_x = 0 \quad \text{at} \quad y = h/2. \tag{38}$$

Substituting the boundary conditions into Equation (36), we get

$$v_x = -\frac{k}{\mu} B \left[1 - \frac{\cosh(y/\sqrt{k})}{\cosh(h/(2\sqrt{k}))} \right]. \tag{39}$$

The fluid flux q is equal to the flow rate per unit cross-sectional area:

$$q = \frac{1}{h} \int_{h/2}^{h/2} v_x dy = -\frac{k}{\mu} B \left[1 - \frac{2\sqrt{k}}{h} \tanh\left(\frac{h}{2\sqrt{k}} \right) \right]. \tag{40}$$

The effective hydraulic conductivity of the channel is defined as the ratio of the fluid flux to the pressure gradient. Thus,

$$H_{channel} = \frac{q}{-B} = \frac{k}{\mu} \left[1 - \frac{2\sqrt{k}}{h} \tanh\left(\frac{h}{2\sqrt{k}} \right) \right]. \tag{41}$$

$H_{channel}$ can be normalized by k/μ, which is the hydraulic conductivity of the porous medium within the channel. Equation (41) indicates that the ratio of $H_{channel}$ to k/μ depends only on $h/\sqrt{k}$. This dependence is plotted in Figure 9. The plot demonstrates that the effect of the channel wall on the fluid flow is less than 10% if $h/\sqrt{k} > 20$. In that case, $H_{channel} \approx k/\mu$ and the momentum equation can be approximated by Darcy's law. From this example, one can observe that $\sqrt{k}$ is a characteristic length scale that determines the validity of Darcy's law.

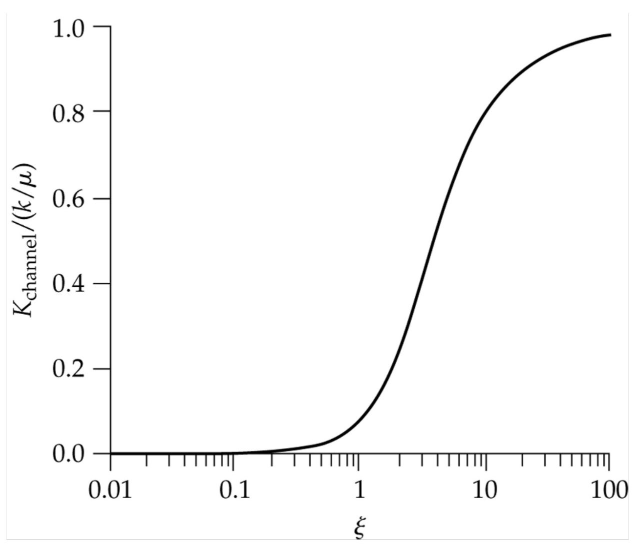


FIG. 9: The dependence of $H_{channel}$ on $\xi = h/\sqrt{k}$.

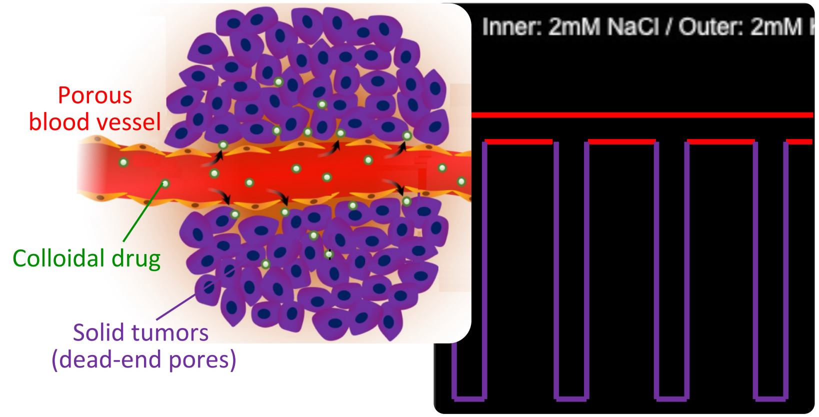
Taylor Hydrodynamic Dispersion & Its Implications on Modern Computation

Transport of particles in fluid is ubiquitous

Environmental sustainability[1]

particle: algae, contaminants,…
fluid: sea water

Engineered drug delivery[2]



particle: blood cells, drug macromolecules,…
fluid: blood

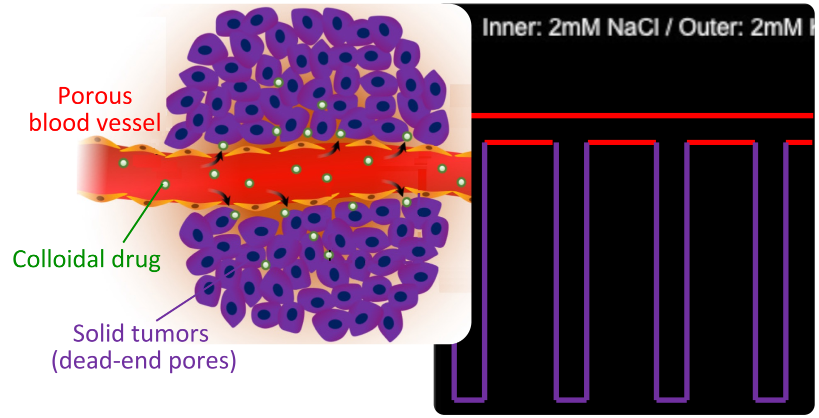
1. Giddings, Talley at UCSD; 2. Shin et al. (2019) *Phys. Rev. Applied*

Transport of particles in fluid is ubiquitous

Environmental sustainability[1]

particle: algae, contaminants,…
fluid: sea water

Engineered drug delivery[2]



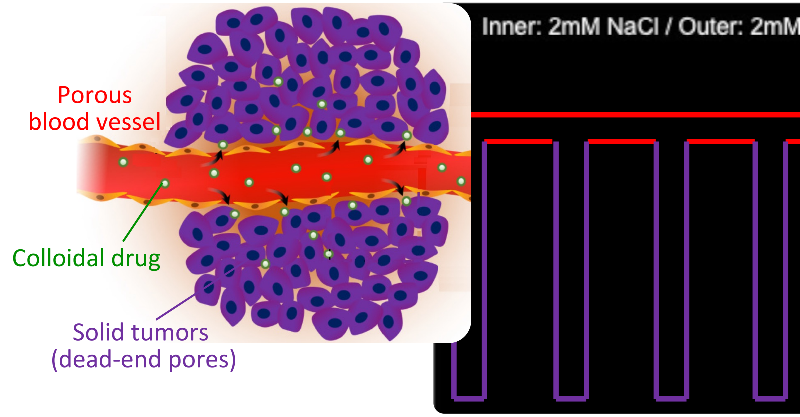
particle: blood cells, drug macromolecules,…
fluid: blood

1. Giddings, Talley at UCSD; 2. Shin et al. (2019) *Phys. Rev. Applied*

Transport of particles in fluid is ubiquitous

Environmental sustainability[1]

Engineered drug delivery[2]



S : particle concentration
u : fluid velocity
[e.g. obtained from Navier-Stokes eq.]
D : particle diffusivity

particles transport

$$\underbrace{\frac{\partial S}{\partial t}}_{\text{temporal}} + \underbrace{\boldsymbol{u}\cdot\nabla S}_{\text{convection}} = \underbrace{D\nabla^2 S}_{\text{diffusion}}$$

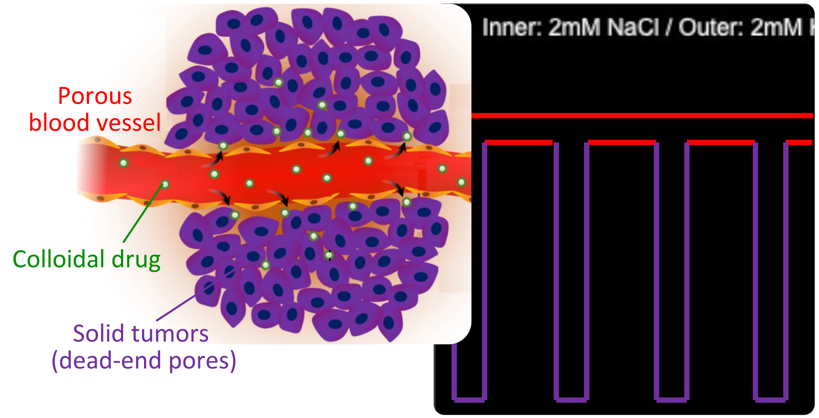
convection ~ translation

1. Giddings, Talley at UCSD; 2. Shin et al. (2019) *Phys. Rev. Applied*

Transport of particles in fluid is ubiquitous

Environmental sustainability[1]

Engineered drug delivery[2]

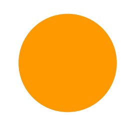


S : particle concentration
u : fluid velocity
[e.g. obtained from Navier-Stokes eq.]
D : particle diffusivity

particles transport

$$\underbrace{\frac{\partial S}{\partial t}}_{\text{temporal}} + \underbrace{\boldsymbol{u} \cdot \nabla S}_{\text{convection}} = \underbrace{D \nabla^2 S}_{\text{diffusion}}$$

diffusion ~ spreading



1. Giddings, Talley at UCSD; 2. Shin et al. (2019) *Phys. Rev. Applied*

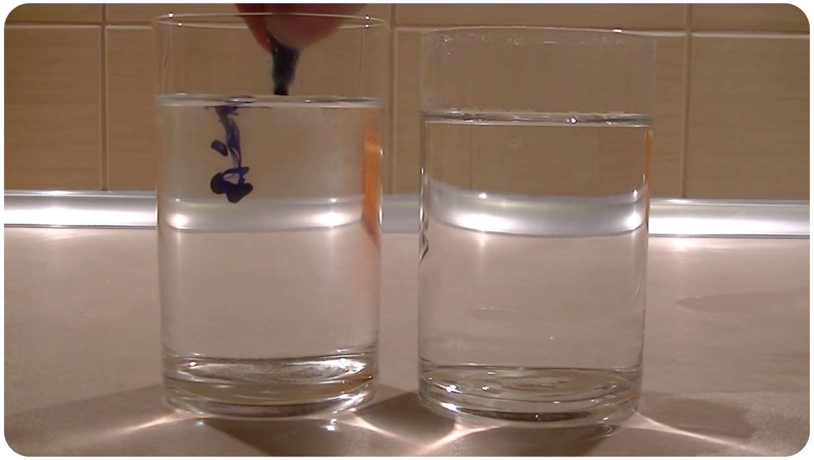
Transport of particles in fluid is ubiquitous

S : particle concentration
u : fluid velocity
[e.g. obtained from Navier-Stokes eq.]
D : particle diffusivity

particles transport

$$\underbrace{\frac{\partial S}{\partial t}}_{\text{temporal}} + \underbrace{\boldsymbol{u}\cdot\nabla S}_{\text{convection}} = \underbrace{D\nabla^2 S}_{\text{diffusion}}$$

high conc. to low conc.



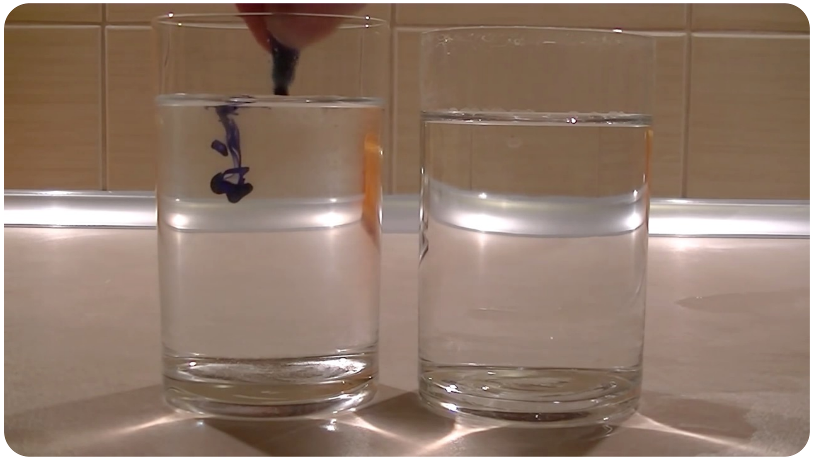
Transport of particles in fluid is ubiquitous

S : particle concentration
u : fluid velocity
[e.g. obtained from Navier-Stokes eq.]
D : particle diffusivity

particles transport

$$\underbrace{\frac{\partial S}{\partial t}}_{\text{temporal}} + \underbrace{\boldsymbol{u} \cdot \nabla S}_{\text{convection}} = \underbrace{D \nabla^2 S}_{\text{diffusion}}$$

high conc. to low conc.



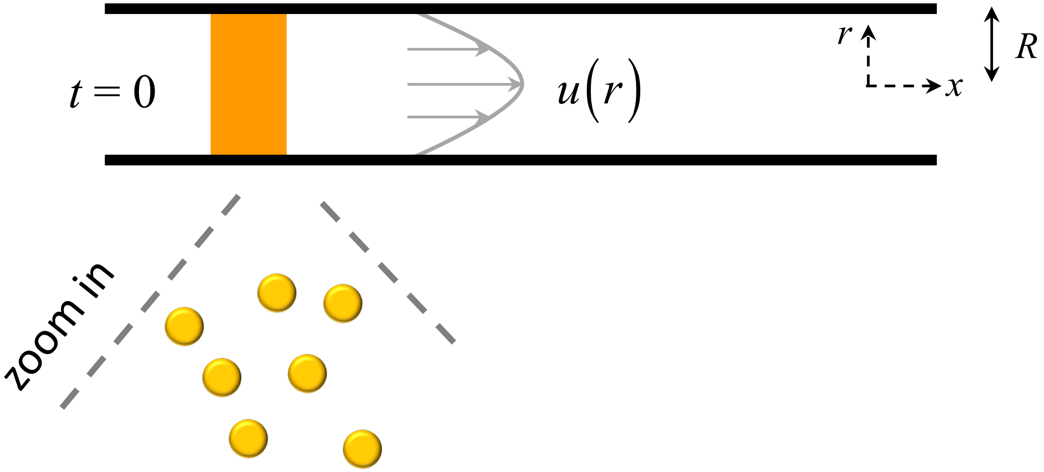
Challenges:

Modeling and predicting particles transport is computationally expensive

Complicated flow fields, complicated geometries, multiple types of particles,…

Simplified model by leveraging the beauty of natural physics!

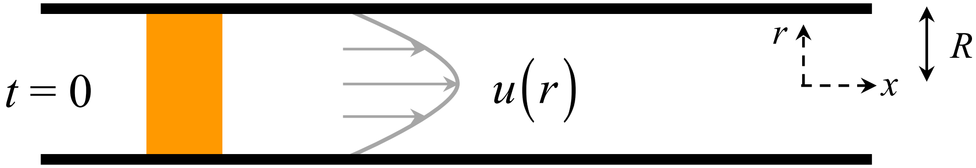
Taylor dispersion: Convection-enhanced diffusion of solute (particles)

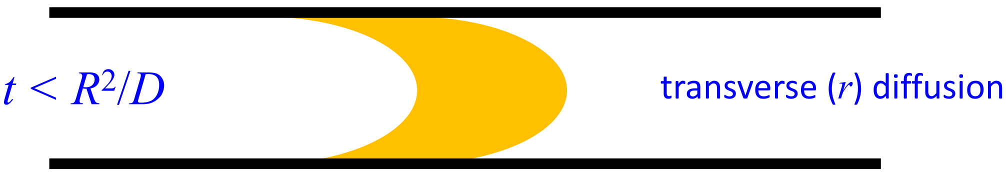


G. I. Taylor

Taylor (1953) *Proc. R. Soc. Lond. A*

Taylor dispersion: Convection-enhanced diffusion of solute (particles)



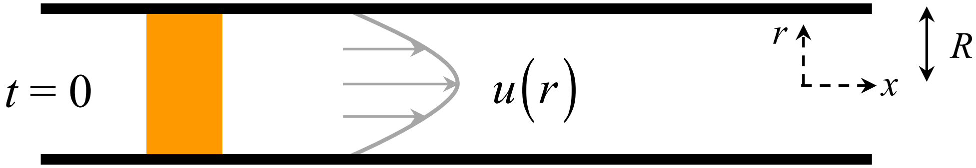


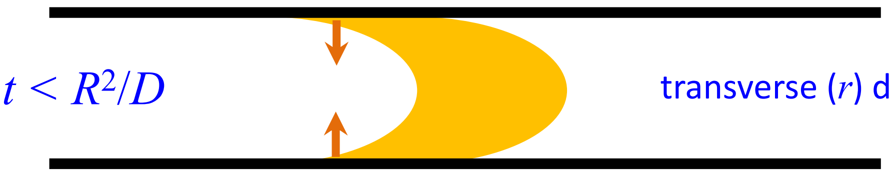
diffusion acts to smooth concentration gradients…

G. I. Taylor

Taylor (1953) *Proc. R. Soc. Lond. A*

Taylor dispersion: Convection-enhanced diffusion of solute (particles)





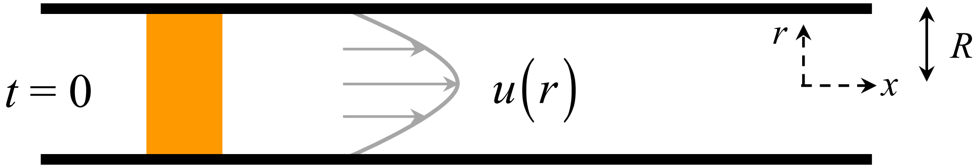
transverse (r) diffusion

G. I. Taylor

diffusion acts to smooth concentration gradients…

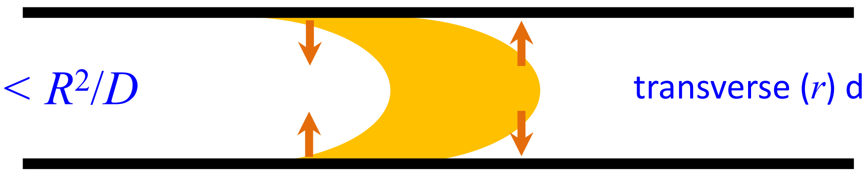
Taylor (1953) *Proc. R. Soc. Lond. A*

Taylor dispersion: Convection-enhanced diffusion of solute (particles)



G. I. Taylor

$t < R^2/D$

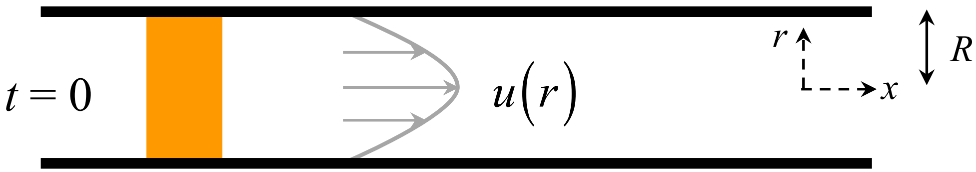


transverse (r) diffusion

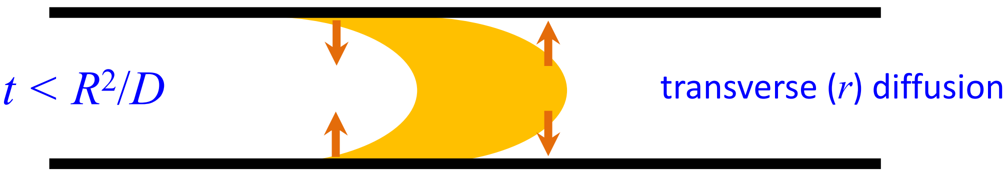
diffusion acts to smooth concentration gradients…

Taylor (1953) *Proc. R. Soc. Lond. A*

Taylor dispersion: Convection-enhanced diffusion of solute (particles)



G. I. Taylor



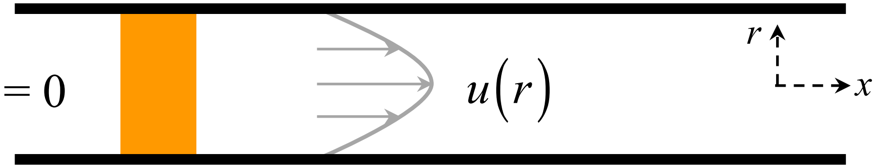
$\sim$ 10 s for R = 100 μm and $D_s = 10^{-9}$ m^2 s^{-1}

…at long times a ***1D* convection-diffusion process**

Taylor (1953) *Proc. R. Soc. Lond. A*

Taylor dispersion: Convection-enhanced diffusion of solute (particles)

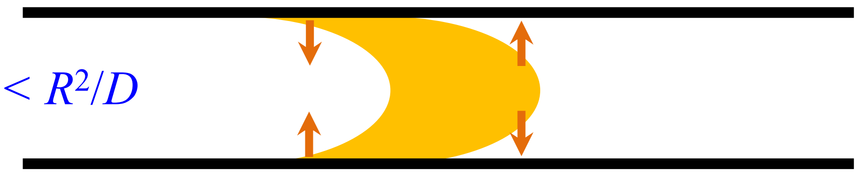
$t = 0$



2D: $[S(r,x)]$

$$\frac{\partial S}{\partial t} + u(r)\frac{\partial S}{\partial x} = \frac{D}{r}\frac{\partial}{\partial r}\left(r\frac{\partial S}{\partial r}\right) + D\frac{\partial^2 S}{\partial x^2}$$

$t < R^2/D$



coupling between convection and transverse diffusion
gives rise to
enhanced axial diffusion (Taylor dispersion)

$t > R^2/D$

$\overline{\bullet}$: cross-sectional average

1D: $[\bar{S}(x)]$

$$\boxed{\frac{\partial \bar{S}}{\partial t} + \bar{u}\frac{\partial \bar{S}}{\partial x} = \left(D + D_T\right)\frac{\partial^2 \bar{S}}{\partial x^2}}$$

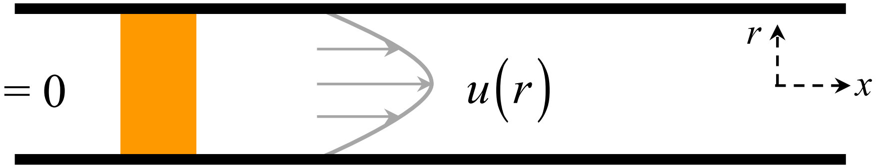
Taylor dispersion (coefficient)

...at long times a ***1D* convection-diffusion process**

Taylor (1953) *Proc. R. Soc. Lond. A*

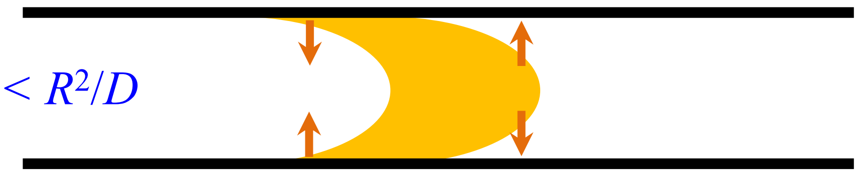
Taylor dispersion: Convection-enhanced diffusion of solute (particles)

$t = 0$



2D: $[S(r,x)]$ $$\frac{\partial S}{\partial t} + u(r)\frac{\partial S}{\partial x} = \frac{D}{r}\frac{\partial}{\partial r}\left(r\frac{\partial S}{\partial r}\right) + D\frac{\partial^2 S}{\partial x^2}$$

$t < R^2/D$



coupling between convection and transverse diffusion
gives rise to
enhanced axial diffusion (Taylor dispersion)

$\overline{\bullet}$: cross-sectional average

$t > R^2/D$

1D: $[S(x)]$ $$\boxed{\frac{\partial \overline{S}}{\partial t} + \overline{u}\frac{\partial \overline{S}}{\partial x} = \left(D + D_T\right)\frac{\partial^2 \overline{S}}{\partial x^2}}$$

Taylor dispersion (coefficient)

...at long times a ***1D* convection-diffusion process**

significantly reducing computational cost
by 100-1000 times!

Taylor (1953) *Proc. R. Soc. Lond. A*

To evaluate Taylor dispersion coefficient

First, we need to know the flow profile:

pressure-driven flow in parallel-plate channel:

$$u(y)=\left(-\frac{dp}{dx}\right)\frac{h^2}{2\mu}\left[\frac{y}{h}-\left(\frac{y}{h}\right)^2\right]$$

pressure-driven flow in a circular tube:

$$u(y)=\left(-\frac{dp}{dx}\right)\frac{R^2}{4\mu}\left[1-\left(\frac{r}{R}\right)^2\right]$$

other flow profiles: Couette flow, flowing down an inclined plane,…

To evaluate Taylor dispersion coefficient

First, we need to know the flow profile:

pressure-driven flow in parallel-plate channel:

$$u(y) = \left(-\frac{dp}{dx}\right)\frac{h^2}{2\mu}\left[\frac{y}{h} - \left(\frac{y}{h}\right)^2\right]$$

pressure-driven flow in a circular tube:

$$u(y) = \left(-\frac{dp}{dx}\right)\frac{R^2}{4\mu}\left[1 - \left(\frac{r}{R}\right)^2\right]$$

other flow profiles: Couette flow, flowing down an inclined plane,…

Second, we solve for Taylor dispersion D_T: $D_T = -\overline{Nu}$ (overbar denotes an average cross the channel cross-section)

where the function N is governed by the following ordinary differential equation:

Governing equation: $D\nabla_i^2 N = u - \overline{u}$ (subscript *i* denotes direction transverse to the unidirectional flow, e.g. *r* in circular channel flow)

Boundary conditions: No flux condition at channel walls and/or symmetry condition about channel centerline

Working examples for computing Taylor dispersion coefficients for steady unidirectional flows

1 Pressure-driven flow in a circular tube

Our goal is to obtain the Taylor dispersion coefficient for a steady, laminar pressure-driven flow in a circular tube. The unidirectional flow is directed along the axial direction x. For a tube of radius R, the velocity and cross-sectionally averaged velocity profiles are, respectively,

$$u(r) = \frac{KR^2}{4\mu}\left[1-\left(\frac{r}{R}\right)^2\right] = 2\overline{u}\left[1-\left(\frac{r}{R}\right)^2\right], \tag{1}$$

$$\overline{u} = \frac{KR^2}{8\mu}, \tag{2}$$

where r is the radial position inside the channel. The cross-sectional average of a quantity is defined by $\overline{(\cdot)} = (1/\pi R^2)\int_0^{2\pi}\int_0^R (\cdot) r dr d\theta$ with θ being the angular position.

The Taylor dispersion coefficient is given by

$$D_T = -\overline{Nu}. \tag{3}$$

To solve for the Taylor dispersion coefficient, first we need to solve the following ordinary differential equation for the parameter N:

$$\text{Governing equation:} \quad D\frac{1}{r}\frac{d}{dr}\left(r\frac{dN}{dr}\right) = u - \overline{u} \tag{4}$$

$$\text{Boundary conditions:} \quad D\frac{dN}{dr} = 0 \quad \text{at} \quad r = 0, R, \tag{5}$$

Integrating (4) once gives

$$\tag{6}$$

where c_1 is an integration constant to be determined. Applying the boundary condition at $r = 0$ gives $c_1 = 0$. Using this result and further integrating (6) once gives

$$\tag{7}$$

where c_2 is an integration constant to be determined. Due to the nature of the ordinary differential equation, the second boundary condition cannot be imposed to determine c_2. However, researchers have found that the second boundary condition can be replaced by

(8)

Applying (8) to (7), we obtain $c_2 = -\overline{u}R^2/12D$. Substituting this result into (7) gives

(9)

Finally, we substitute (1) and (9) into (3). After some algebra, we obtain the Taylor dispersion coefficient as

(10)

2 Pressure-driven flow in a parallel-plate channel

Our goal is to obtain the Taylor dispersion coefficient for a steady, laminar pressure-driven flow in a parallel-plate channel. The unidirectional flow is directed along the axial direction x. For a channel of height h, the velocity and cross-sectionally averaged velocity profiles are, respectively,

$$u(y) = \frac{Kh^2}{2\mu}\left[\frac{y}{h} - \left(\frac{y}{h}\right)^2\right] = 6\overline{u}\left[\frac{y}{h} - \left(\frac{y}{h}\right)^2\right], \tag{11}$$

$$\overline{u} = \frac{Kh^2}{12\mu}, \tag{12}$$

where $K = -dp/dx$ is the axial pressure gradient, y is the direction perpendicular to the channel walls, and μ is the fluid dynamic viscosity. The cross-sectional average of a quantity is defined by $\overline{(\cdot)} = (1/h)\int_0^h (\cdot)dy$.

As we discussed in class, the Taylor dispersion coefficient is given by

$$D_T = -\overline{Nu}. \tag{13}$$

To solve for the Taylor dispersion coefficient, first we need to solve the following ordinary differential equation for the parameter N:

$$\text{Governing equation:} \quad D\frac{d^2N}{dy^2} = u - \overline{u} \tag{14}$$

$$\text{Boundary conditions:} \quad D\frac{dN}{dy} = 0 \quad \text{at} \quad y = 0, h, \tag{15}$$

where D is the intrinsic diffusivity of the species. Integrating (14) once gives

(16)

where c_1 is an integration constant to be determined. Applying the boundary condition at $y = 0$ gives $c_1 = 0$. Using this result and further integrating (16) once gives

(17)

where c_2 is an integration constant to be determined. Due to the nature of the ordinary differential equation, the second boundary condition cannot be imposed to determine c_2. However, researchers have found that the second boundary condition can be replaced by

(18)

Applying (18) to (17), we obtain $c_2 = \overline{u}h^2/60D$. Substituting this result into (17) gives

(19)

Finally, we substitute (11) and (19) into (13). After some algebra, we obtain the Taylor dispersion coefficient as

(20)