UNIT-3

DYNAMICS

Elementary Fluid Dynamics

- Understanding the physics of ***fluid in motion***
- Derivation of the ***Bernoulli equation*** from Newton's second law

⦿ Basic Assumptions of fluid stream, unless a specific comment

1st assumption: ***Inviscid fluid***(Zero viscosity = Zero shearing stress)

$\rightarrow$ No force by wall of container and boundary

$\rightarrow$ Applied force = Only Gravity + Pressure force

※ Newton's Second Law of Motion of a ***Fluid Particle***

(Net pressure force) + (Gravity)

(Fluid mass) $\times$ (Acceleration)

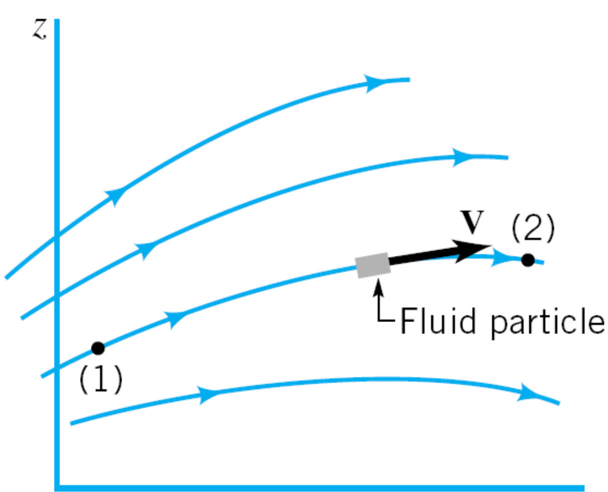
2nd assumption:*Steady flow* (?)

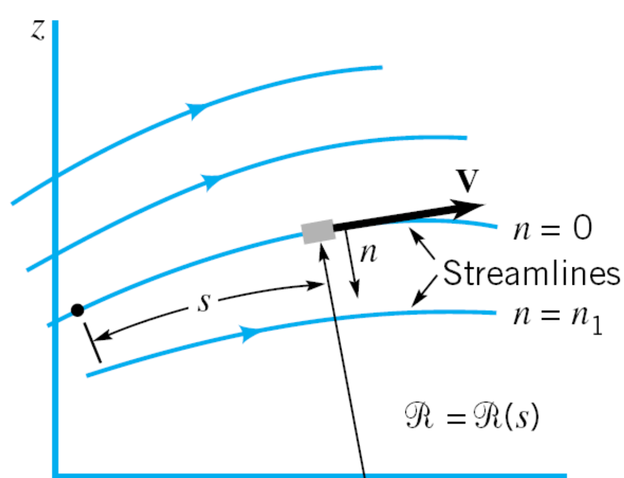
$\rightarrow$ No Change of flowing feature with time at a given location

$\rightarrow$ Every successive particle passing though the same point

: Same path (called ***streamline***) &

Same velocity (***tangential*** to the streamline)





⦿ Additional Basic Terms in Analysis of Fluid Motion

•***Streamline*** (Path of a fluid particle)

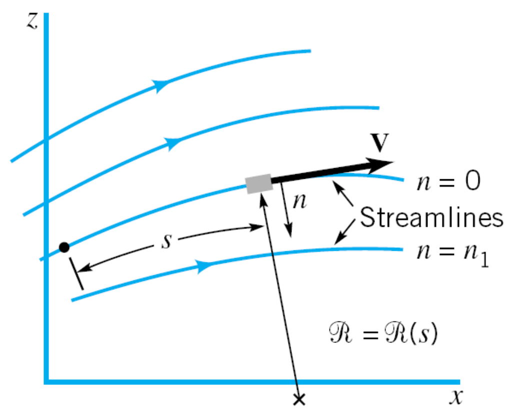
- Position of a particle

$$= f(r_o, \vec{v})$$

where r_o : Initial position,

$\vec{v}$: Velocity of particle

- No streamlines intersecting each other



• Two Components in ***Streamline Coordinates*** (See the figure)

1. ***Tangential*** coordinate: $s = s(t)$

: ***Moving distance*** along streamline,

: Related to Particle's speed $(v = ds/dt)$

2. ***Normal*** coordinate: $n = n(t)$

: ***Local radius of curvature*** of streamline $R = R(s)$

: Related to Shape of the streamline

• Two Accelerations of a fluid particle along ***s*** and ***n*** coordinates

1. Streamwise acceleration (☞ Change of the speed)

$$a_s = \frac{dv}{dt} = \frac{\partial v}{\partial s}\frac{ds}{dt} = \frac{\partial v}{\partial s} v$$ using the Chain rule

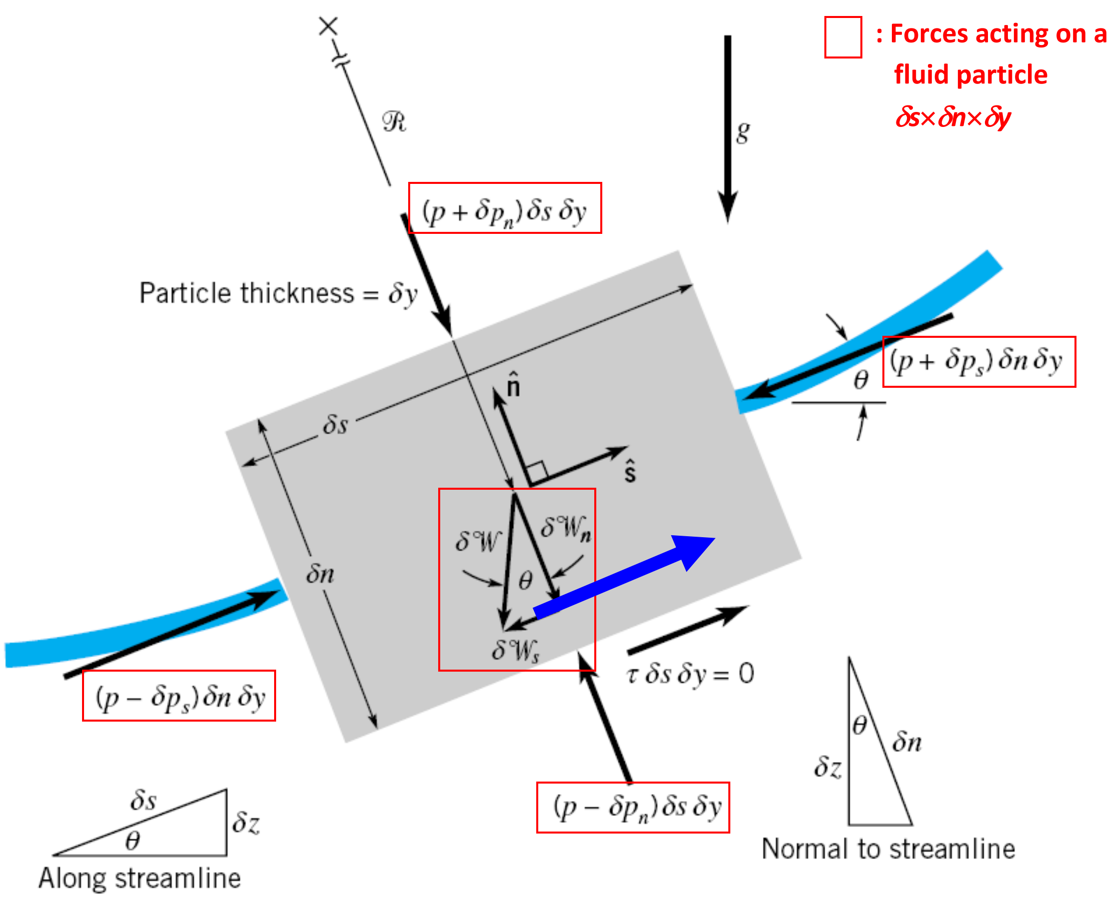
2. Normal acceleration (☞ Change of the direction)

$$\boxed{a_n = \frac{v^2}{R}}$$ (: Centrifugal acceleration)

Q. What generate these a_s and a_n? (***Pressure force*** and***Gravity***)

Part 1. Newton's second law ***along a streamline*** ($\hat{s}$ direction)

Consider a small fluid particle of size $\delta s \times \delta n \times \delta y$ as shown



Newton's second law in $\hat{s}$ direction

$$\sum \delta F_s = \delta m a_s = \delta m v \frac{\partial v}{\partial s} = \boxed{\rho \delta \forall v \frac{\partial v}{\partial s}} \qquad \text{along } \hat{s} \text{ direction}$$

= Gravity force + Net Pressure force

where $\delta \forall$: Volume of a fluid particle = $\delta s \times \delta n \times \delta y$

(i) ***Gravity force*** along $\hat{s}$ direction

$$\delta W_s = -\delta W \sin\theta = \boxed{-(\gamma \delta \forall)\sin\theta}$$

(ii) ***Pressure force*** along $\hat{s}$ direction

Let p: Pressure at the center of $\delta \forall$

$p - \dfrac{\partial p}{\partial s}\dfrac{\delta s}{2}$: Average pressures at Left face (Decrease)

$p + \dfrac{\partial p}{\partial s}\dfrac{\delta s}{2}$: Average pressures at Right face (Increase)

Then, ***Net pressure force*** along $\hat{s}$ direction, $\delta F_{ps} =$ (Pressure)×(Area)

$$\delta F_{ps} = (p - \frac{\partial p}{\partial s}\frac{\delta s}{2})\delta n \delta y - (p + \frac{\partial p}{\partial s}\frac{\delta s}{2})\delta n \delta y = -\frac{\partial p}{\partial s}\delta s \delta n \delta y = \boxed{-\frac{\partial p}{\partial s}\delta V\!\!\!-}$$

: Depends not on p itself,but on $\frac{\partial p}{\partial s}$ (***Rate of change inp***)

• Total force in $\hat{s}$ direction (Streamline)

$$\sum \delta F_s = \delta W_s + \delta F_{ps}$$

☞ $$\rho \delta V\!\!\!- a_s = \rho \delta V\!\!\!- v \frac{\partial v}{\partial s} = (-\gamma \sin\theta - \frac{\partial p}{\partial s})\delta V\!\!\!-$$

Finally, Newton's second law ***along a streamline*** ($\hat{s}$ direction)

$$\therefore \boxed{\rho a_s = \rho v \frac{\partial v}{\partial s} = -\gamma \sin\theta - \frac{\partial p}{\partial s}}$$

Change of Particle's speed

☞ Affected by Weight and Pressure Change

※ Making this equation more familiar

$$\rho v \frac{\partial v}{\partial s} = -\gamma \sin\theta - \frac{\partial p}{\partial s}$$ ☞ $$\boxed{\frac{1}{2}\rho \frac{d(v^2)}{ds} = -\gamma \frac{dz}{ds} - \frac{dp}{ds}}$$

because $\sin\theta = \frac{dz}{ds}$ (See the figure above)

$$\frac{\partial p}{\partial s} = \frac{dp}{ds} \quad \text{using } dp = \frac{\partial p}{\partial s} ds + \frac{\partial p}{\partial n} dn = \frac{\partial p}{\partial s} ds \quad \text{(why?)}$$

$$v\frac{\partial v}{\partial s} = v\frac{dv}{ds} = \frac{1}{2}\frac{dv^2}{ds}$$

or $$dp + \frac{1}{2}\rho d(v^2) + \gamma dz = 0$$ (Divided by *ds*)

or $$\int \frac{dp}{\rho} + \frac{1}{2}v^2 + gz = \text{constant}$$ (By integration)

By assuming a constant ρ (Incompressible fluid): **3rd assumption**

$$\therefore \boxed{p + \frac{1}{2}\rho v^2 + \gamma z = \text{Constant}}$$ along streamline ($\hat{s}$ direction)

: ***Bernoulli equation along a streamline***

☞Valid for (1) a steady flow of (2) incompressible fluid

(3) without shearing stress

c.f. If ρ is not constant (Compressible, **e.g**. Gases),

☞ $\rho = \rho(p)$: Must be known to integrate $\int \frac{dp}{\rho}$.

⦿ **What this *Bernoulli* Equation means? (Physical Interpretation)**

For a ***Steady*** flow of ***Inviscid*** and ***Incompressible*** fluid,

$$p + \frac{1}{2}\rho v^2 + \gamma z = \text{Constant} \qquad \text{along streamline} \qquad (1)$$

: Mathematical statements of ***Work-energy principle***

• Unit of Eq. (1): [N/m^2] = [N·m/m^3] = [Energy per unit volume]

p = Works on unit fluid volume done by pressure

γz = Works on unit fluid volume done by weight

$\frac{1}{2}\rho v^2 =$ Kinetic energy per unit fluid volume

※ Same ***Bernoulli Equations in different units***

1. Eq (1) $\div \gamma$ ☞ [N·m/m^3] $\div$[N·m/m^3] = [m] = [**Length unit**]

$$\boxed{\frac{p}{\gamma} + \frac{v^2}{2g} + z = \text{Constant}} \qquad \text{(Head unit)}$$

$\frac{p}{\gamma}$: Depth of a fluid column produce p (***Pressure head***)

$\dfrac{v^2}{2g}$: Height of a fluid particle to reach v from rest by free falling

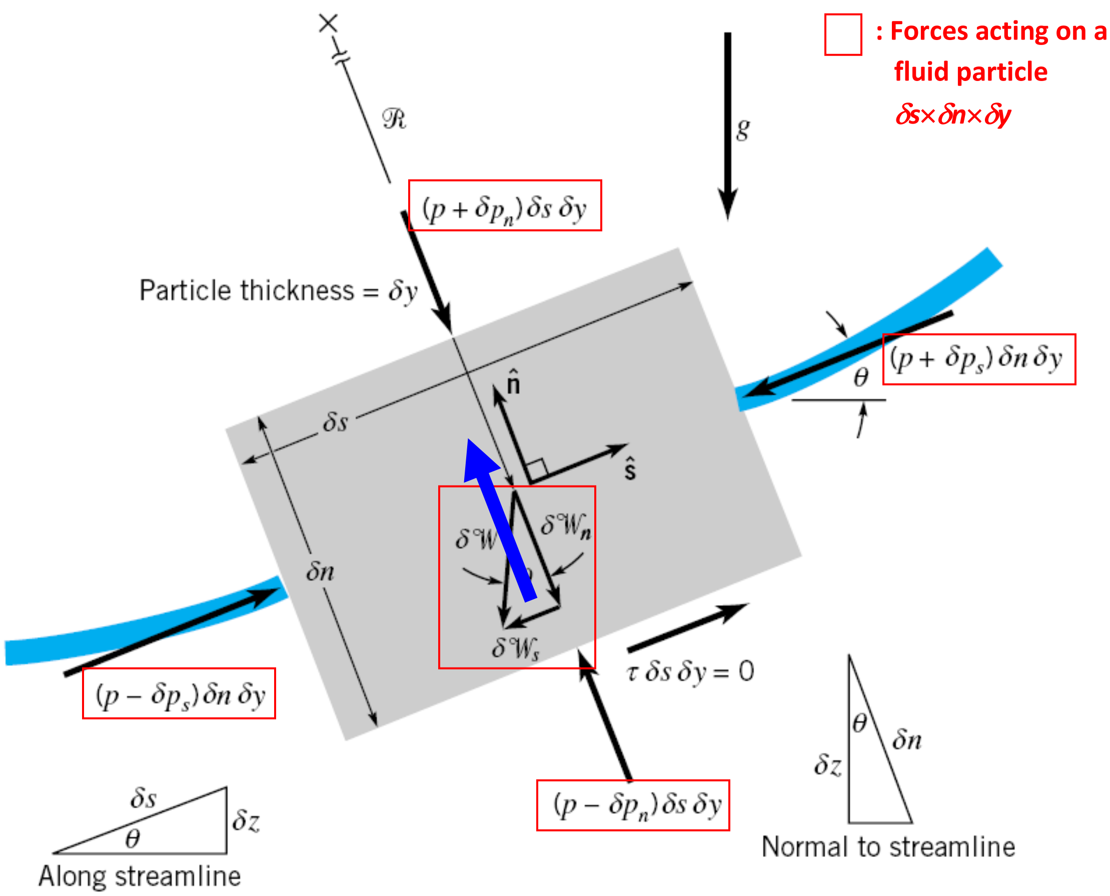
(***Velocity head***)

z: Height corresponding to Gravitational potential (***Elevation head***)

Part 2. Newton's second law ***normal to a streamline*** ($\hat{n}$ direction)

Consider the same situation as Sec. 3.3 shown in figure

For a small fluid particle of size $\delta s \times \delta n \times \delta y$ as shown



Newton’s second law in $\hat{n}$ direction

$$\sum \delta F_n = \delta m a_n = \delta m \frac{v^2}{R} = \boxed{\rho \delta \cancel{V} \frac{v^2}{R}} \qquad \text{along } \hat{n} \text{ direction}$$

= Gravity force + Net Pressure force

(i) ***Gravity force*** along $\hat{n}$ direction

$$\delta W_n = -\delta W \cos\theta = \boxed{-(\gamma \delta \cancel{V}) \cos\theta}$$

(ii) ***Pressure force*** along $\hat{n}$ direction

By the same manner in the previous case,

$$\delta F_{pn} = \left(p - \frac{\partial p}{\partial n}\frac{\delta n}{2}\right)\delta s \delta y - \left(p + \frac{\partial p}{\partial n}\frac{\delta n}{2}\right)\delta s \delta y = -\frac{\partial p}{\partial n}\delta n \delta s \delta y = \boxed{-\frac{\partial p}{\partial n}\delta \cancel{V}}$$

- Total force in $\hat{n}$ direction (Normal to Streamline)

$$\sum \delta F_n = \delta W_n + \delta F_{pn}$$

☞ $$\rho\delta\cancel{V}a_n = \rho\delta\cancel{V}\frac{v^2}{R} = (-\gamma\cos\theta - \frac{\partial p}{\partial n})\delta\cancel{V}$$

$\therefore \boxed{\rho\frac{v^2}{R} = -\gamma\cos\theta - \frac{\partial p}{\partial n}}$ *normal to streamline*($\hat{n}$ direction)

Change of Particle's direction of motion

☞ Affected by Weight and Pressure Change along $\hat{n}$

Ex. If a fluid flow: ***Steep direction change*** ($R\downarrow$) or ***fast flow*** ($v\uparrow$)

or ***heavy*** ($\rho\uparrow$) fluid

☞ Generate large force unbalance

- Special case: Standing close to a Tornado

i.e. Gas flow (Negligible γ □) in horizontal motion ($\frac{dz}{dn}$ = 0)

$$-\gamma\cancel{\frac{dz}{dn}} - \frac{\partial p}{\partial n} = \rho\frac{v^2}{R} \quad \rightarrow \quad \frac{\partial p}{\partial n} = -\rho\frac{v^2}{R} < 0$$ (Attractive)

: Moving closer ($R\downarrow$) ☞ More dangerous ($\frac{\partial p}{\partial n}\uparrow$)

※ Making this equation more familiar

By the same manner as the previous case,

$$\rho\frac{v^2}{R} = -\gamma\cos\theta - \frac{\partial p}{\partial n} ☞ \rho\frac{v^2}{R} = -\gamma\frac{dz}{dn} - \frac{dp}{dn}$$

because $\cos\theta = \dfrac{dz}{dn}$ (See the figure)

$$\frac{\partial p}{\partial n} = \frac{dp}{dn} \text{ , since } dp = \frac{\partial p}{\partial s} ds + \cancel{\frac{\partial p}{\partial n}} dn = \frac{\partial p}{\partial n} dn$$

or $\displaystyle\int \frac{dp}{\rho} + \int \frac{v^2}{R} dn + gz$ = Constant (normal to streamline)

By assuming a constant ρ (Incompressible fluid): **3rd assumption**

$$\therefore \boxed{p + \rho \int \frac{v^2}{R} dn + \gamma z = \text{Constant}} \qquad \text{(normal to streamline)}$$

: ***Bernoulli equation normal to streamline*** ($\hat{n}$ direction)

☞Valid for (1) a steady flow of (2) incompressible fluid

(3) without shearing stress

Ch4 Fluid Kinematics

In Ch1-3: Fluid at rest (stationary or moving) in a rather elementary manner.

Real fluids: slightly viscous shear and pressure will cause fluid to deform and move

Objectives: The kinematics of the fluid motion
—the velocity
—acceleration, and

—the description and visualization of its motion

(The dynamics of the motion—The analysis of the specific forces necessary to produce the motion.)

Examples: Chimney; atmosphere; waves on lake; mixing of paint in a budset…etc.

4.1 The Velocity Field

- Continuum Hypothesis: Fluid particle not molecular
- Field Representation: The representation of fluid parameters as a function of the spatial coordinate (x, y, z, or r,θ, z or r, θ,φ)
- Time (t)

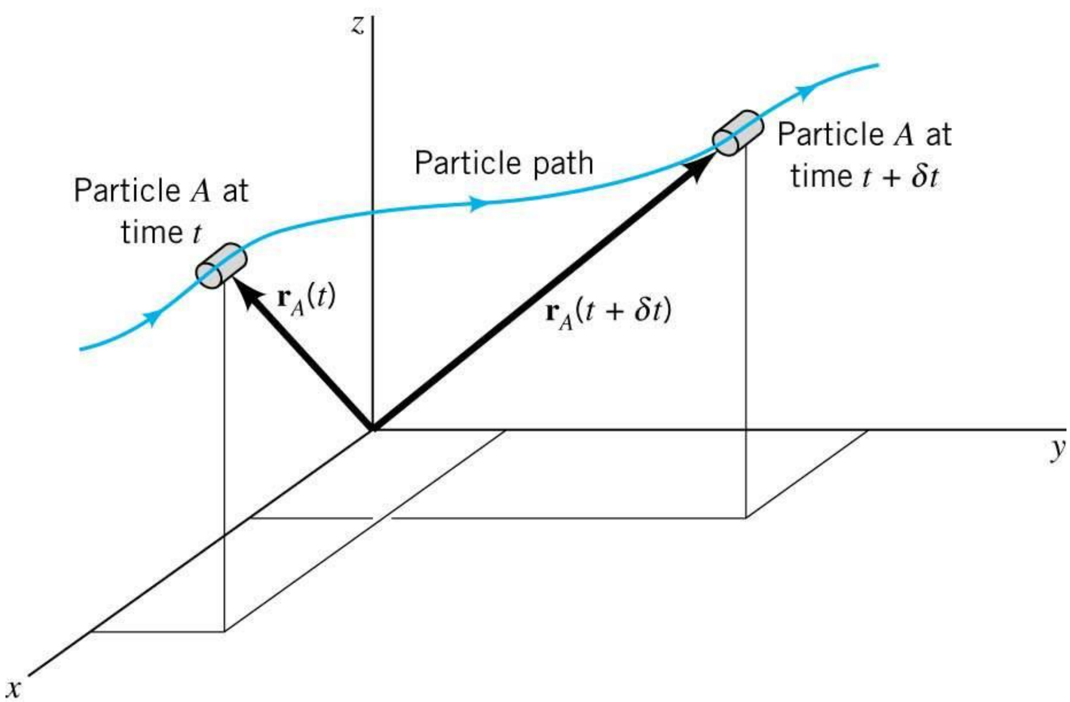
$$\vec{V} = \vec{u}_i(x,y,z,t) = u(x,y,z,t)\vec{i} + v(x,y,z,t)\vec{j} + w(x,y,z,t)\vec{k}$$

where i =1, 2, 3 representing x, y, z, respectively.

$u_1=u$; $u_2=v$; $u_3=w$

$\vec{V} = \vec{V}(x,y,z,t)$ direction & magnitude

Magnitude $\left|\vec{V}\right| = \left(u^2 + v^2 + w^2\right)^{\frac{1}{2}}$



4.1.1 (a) Eluerian description

The velocities (or pressure, density, temperature etc.) are given at fixed points in space V (x_i, t) at which time varies. This corresponds to the usual experimental arrangement where the measuring devices are fixed and the frame of reference is fixed with them. Common and easier to use. However, it would in some ways be better to follow a particle and see what happens near it as it moves along. For instance, in the atmosphere it is the history of mass of air as it moves along that determines whether it will become a shower, rather than the sequence of air masses that pass a weather station (though they one of course related). This leads to

4.1.1(b) Lagrangian description

Here quantities are given for a fixed particle at varying time, so that the velocity is $\vec{V_i}(\underline{x_i^\circ},t)$, where $\underline{r^\circ}$ was the particle's position at t=0.

Unfortunately, the mathematics of the Lagrangian description is hard; but it is often useful to consider the particle's life history in order to gain an understanding of a flows. Lagrangian histories can be obtained in the atmosphere from balloon flights, or in the Gulf Stream from just buoyant devices.

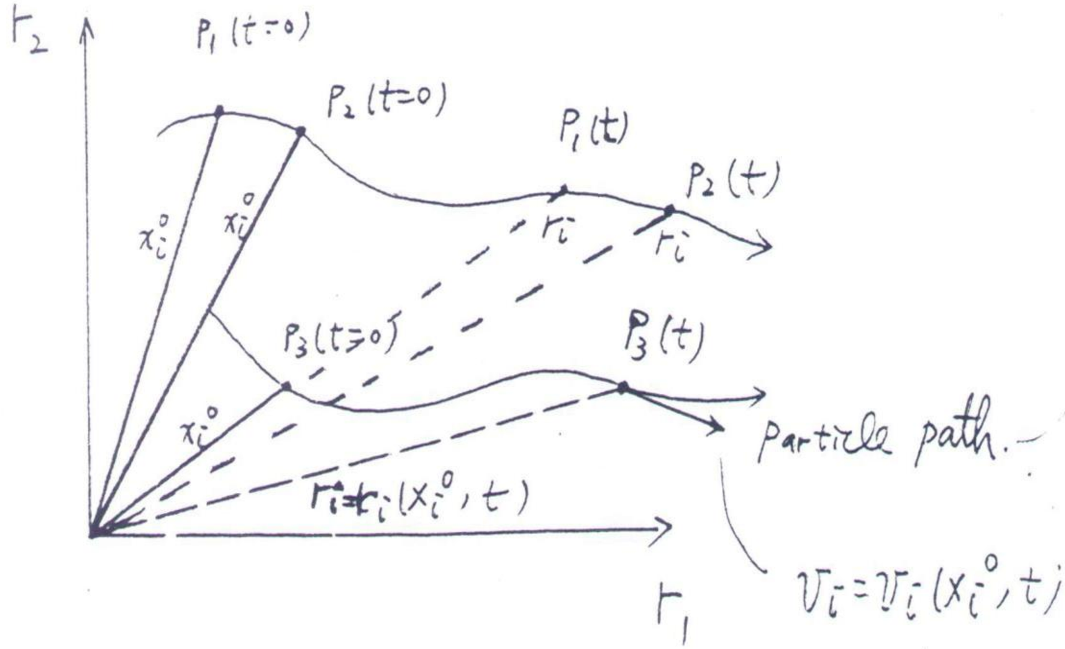
Independent variables in the Lagrangian view point one the initial position x_i° and time t.

Let us use r_i for the position of a material point, or fluid particle. Initially the fluid particle is at the position x_i°, and the particle path through space is given in the form

$$r_i = r_i(x_i^\circ, t);$$

$$V_i = \frac{\partial r_i}{\partial t} \quad \text{and} \quad a_i = \frac{\partial^2 r_i}{\partial t^2}$$

In the Lagrangian description these quantities are functions of particle identification tag x_i° and the time t as shown in the Fig. below.



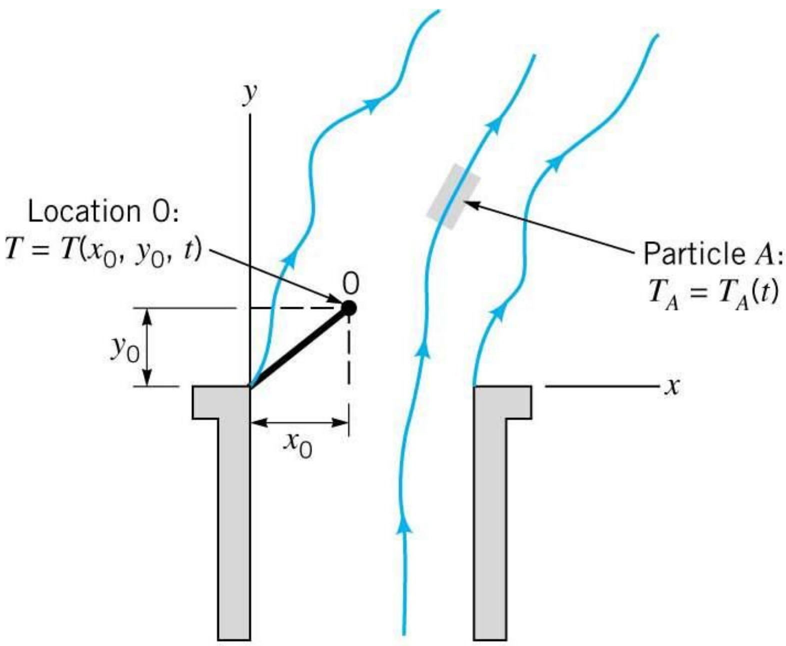


FIGURE 4.2 Eulerian and Lagrangian descriptions of temperature of a flowing fluid.
Examples of using Lagrangian description:

- Atmosphere
- Oceanography
- Bioscience
- Fluid Machinery

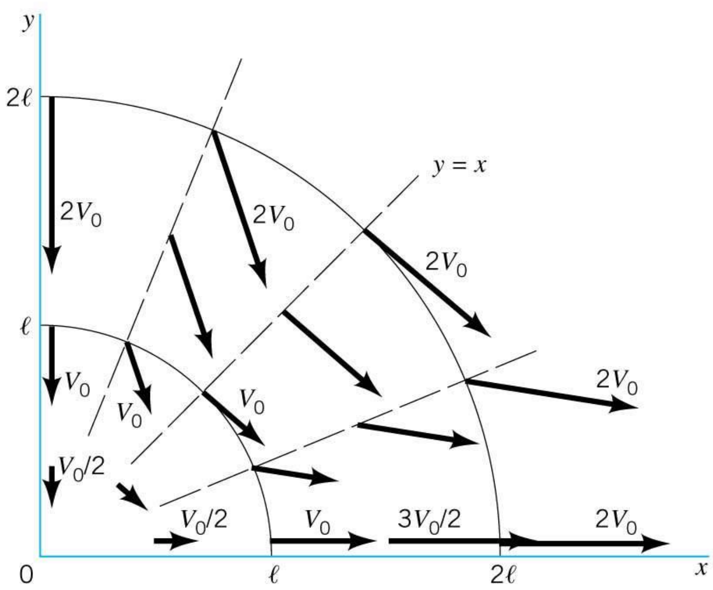
Example 4.1 A velocity field is given by $V=(V_0/l)(x_i-y_j)$, where V_0 and l are constant. At what location in the flow field is the speed equal to V_0? Make a sketch of the velocity field in the first quadrant ($x>=0$, $y>=0$) by drawing arrows representing the fluid velocity at representative locations.

Solution: The x, y, and z components of the velocity are given by $u=V_0x/l$, $v=-V_0y/l$, and $w=0$ so that the fluid speed, V, is

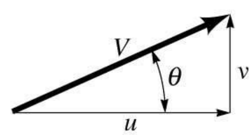
$$V=(u^2+v^2+w^2)^{\frac{1}{2}}=\frac{V_0}{l}(x^2+y^2)^{\frac{1}{2}} \qquad (1)$$

The speed is $V=V_0$ at any location on the circle of radius l centered at the origin $[(x^2+y^2)^{\frac{1}{2}}=l]$ as shown in Fig. E4.1a. (Ans).

The direction of the fluid velocity relative to the x axis is given in terms of $\theta=\arctan(v/u)$ as shown in Fig. E4.1b. For this flow, $tan\,\theta=\frac{v}{u}=\frac{\frac{-V_0y}{l}}{\frac{V_0x}{l}}=\frac{-y}{x}$.



(a)



(b)

4.1.2 1-D, 2-D, and 3-D flows

Generally, a fluid flow (real flow) is 3-D, time-dependent flow.

$$\vec{V} = \vec{V}(x, y, z, t)$$

Simplifying 1D or 2D (one or two of the velocity components) may be small compared to the other.

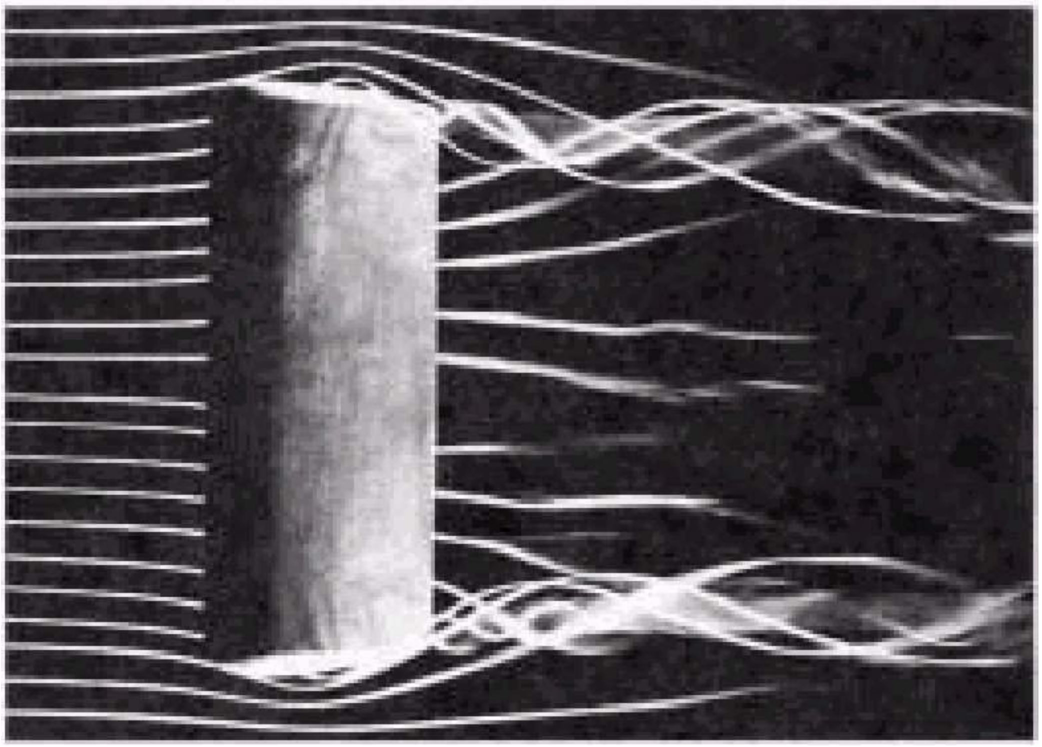


FIG.4.3 **Flow visualization of the complex three-dimensional flow past a model airfoil.**

4.1.3 Steady & Unsteady flow

Steady $\dfrac{\partial \vec{V}}{\partial t} = 0$; almost all flows are unsteady.

Unsteady flows: (a) Non-periodic; (b) Periodic; (c) Random.

Examples: Faucet (loud banging of pipes); Air-gasoline injection

4.1.4 Streamlines, Streaklines, and Pathlines

<u>Streamlines</u>: A line that is everywhere tangent to the velocity field (analytical work).
Steady: fixed lines in space; unsteady: lines may change shape.

2-D flow: $\dfrac{dy}{dx} = \dfrac{v}{u}$

Streaklines: A line that consist of all particles in a flow that have previously passed through a common point.

Pathlines: A line traced out by a given particle as it flows from one point to another (Lagrangian concept).

For steady flow, all three lines are coincide.

Example 4.2: Determine the streamlines for the 2-D steady flow discussed in example 4.1, $V=(V_0/l)(x\mathbf{i}-y\mathbf{j})$.

Solution: Since and it follows that streamlines are given by solution of the equation

$$\frac{dy}{dx}=\frac{v}{u}=\frac{-(V_0/l)y}{(V_0/l)x}=\frac{-y}{x}$$

in which variables can be separated and the equation integrated to give $\int\frac{dy}{y}=-\int\frac{dx}{x}$ or ***ln y = - ln x* + constant**

Thus, along the streamline $xy = C$, where C is a constant.

By using different values of the constant C, we can plot various lines in the x-y plane—the streamlines. The usual notation for a streamline is ψ= constant on a streamline. Thus, the equation for the streamlines of this flow are $\psi = xy$.

As is discussed more fully in Chapter 6, the function $\psi=\psi(x,y)$ is called the *stream function*. The streamlines in the first quadrant are plotted in Fig. E4.2. A comparison of this figure with Fig. E4.1a illustrates the fact that streamlines are lines parallel to the velocity field.

Streamlines can be obtained analytically by integrating the equations defining lines tangent to the velocity field.

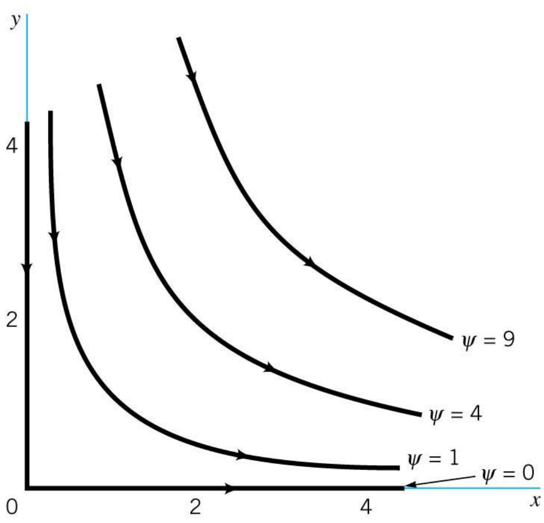


FIGURE E4.2

Example. 4.3 Water flowing from the oscillating slit shown in Fig. E4.3a produces a velocity field given by

$V = u_0 \sin[w(t - y/v_0)]i + v_0 j$, where u_0, v_0, and w are constants. Thus, the y component of velocity remains constant $(v = v_0)$ and the x component of velocity at $y = 0$ coincides with the velocity of the oscillating sprinkler head [$u = u_0 \sin(wt)$ at $y = 0$].

(a) Determine the streamline that passes through the origin at $t = 0$; at $t = \pi/2\omega$. (b) Determine the pathline of the particle that was at the origin at $t = 0$: at $t = \pi/2\omega$. (c) Discuss the shape of the streakline that passes through the origin.

Solution:

(a) Since $u = u_0 \sin[w(t - y/v_0)]$ and $v = v_0$ it follows from Eq. 4.1 that streamlines are given by the solution of

$$\frac{dy}{dx} = \frac{v}{u} = \frac{v_0}{u_0 \sin[w(t - y/v_0)]}$$

in which the variables can be separated and the equation integrated (for any given time t) to give

$$u_0 \int \sin[w(t - \frac{y}{v_0})] dy = v_0 \int dx$$

or

$$u_0 (v_0 / w) \cos[w(t - \frac{y}{v_0})] = v_0 x + C \tag{1}$$

where C is a constant. For the streamline at $t = 0$ that passes the origin ($x = y = 0$), the value of C is obtained from Eq. 1 as $C = u_0 v_0 / w$. Hence, the equation for this streamline is

$$x = \frac{u_0}{w}[\cos(\frac{wy}{v_0}) - 1] \tag{2}$$

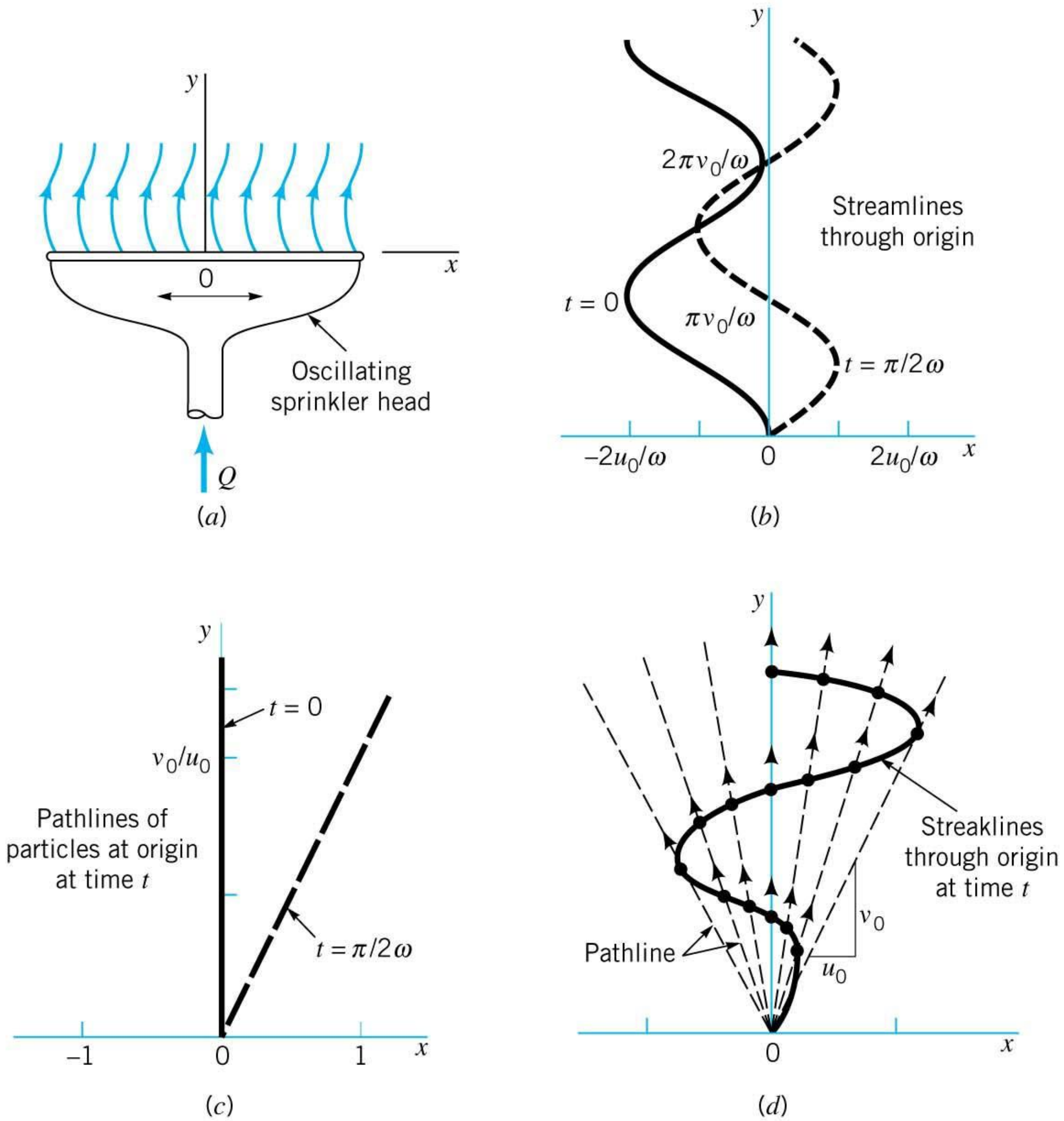


FIGURE E4.3

Similarity, for the streamline at $t = \pi/2\omega$ that passes through the origin. Eq. 1 gives $C = 0$. Thus, the equation for this streamline is

$$x = \frac{u_0}{w}\cos[w(\frac{\pi}{2w} - \frac{y}{v_0})] = \frac{u_0}{w}\cos(\frac{\pi}{2} - \frac{wy}{v_0})$$

Or $$x = \frac{u_0}{w}\sin(\frac{wy}{v_0}) \qquad (3)\ \text{(Ans)}$$

These two streamlines, plotted in Fig. E4.3b, are not the same because the flow is unsteady. For example, at the origin ($x = y = 0$) the velocity is $V = v_0 j$ at $t = 0$ and $V = u_0 i + v_0 j$ at $t = \pi/2\omega$. Thus, the angle of the streamline passing through the origin changes with time. Similarity, the shape of the entire streamline is a function of time.

(b) The pathline of a particle (the location of the particle as a function of time) can be obtained from the velocity field and the definition of the velocity. Since $u = dx/dt$ and v - dy/dt we obtained

$$\frac{dx}{dt} = u_0 \sin[w(t - \frac{y}{v_0})] \qquad \text{and} \qquad \frac{dy}{dt} = v_0$$

The y equation can be integrated (since v_0 = constant) to give the y coordinate of the pathline as $\boldsymbol{y = v_0 t + C_1}$ (4)

where C_1 is a constant. With this known $y = y(t)$ dependence, the x equation for the pathline becomes

$$\frac{dx}{dt} = u_0 \sin[w(t - \frac{v_0 t + C_1}{v_0})] = -u_0 \sin(\frac{C_1 w}{v_0})$$

This can be integrated to give the x component of the pathline as

$$x = -[u_0 \sin(\frac{C_1 w}{v_0})]t + C_2 \qquad (5)$$

where C_2 is a constant. For the particle that was at the origin ($x = y = 0$) at time $t = 0$, Eqs. 4 and 5 give $C_1 = C_2 = 0$. Thus, the pathline is

$$\boldsymbol{x = 0 \text{ and } y = v_0 t} \qquad (6) \quad \text{(Ans)}$$

Similarity, for the particle that was at the origin at $t = \pi/2\omega$. Eqs. 4 and 5 give $C_1 = -\pi v_0/2\omega$. Thus, the pathline for the particle is

$$x = u_0(t - \frac{\pi}{2w}) \ \& \ y = v_0(t - \frac{\pi}{2w}) \qquad (7)$$

The pathline can be drawn by plotting the locus of $x(t)$, $y(t)$ values for $t \geq 0$ or by eliminating the parameter t from Eq. 7 to give

$$y = \frac{v_0}{u_0} x \qquad (8)$$

(Ans)

The pathlines given by Eqs. 6 and 8, shown in Fig. 4.3*c*. are straight lines from the origin (rays). The pathlines and streamlines do not coincide because the flow is unsteady.

(c) The streakline through the origin at time $t = 0$ is the locus of particles at $t = 0$ that previously ($t < 0$) passed through the origin. The general shape of the streaklines can be seen as follows. Each particle that flows through the origin travels in a straight line (pathlines are rays from the origin), the slope of which lies between $\pm v_0/u_0$ as shown in Fig. E4.3*d*. Particles passing through the origin at different times are located on different rays from the origin and at different distances from the origin. The net result is that a stream of dye continually injected at the origin (a streakline) would have the shape shown in Fig. E4.3*d*. Because of the unsteadiness, the streakline will vary with time, although it will always have the oscillating, sinous character shown. Similar streaklines are given by the stream of water from a garden hose nozzle that oscillates back and forth in a direction normal to the axis of the nozzle. In this example neither the streamlines, pathlines, nor streaklines coincide. If the flow were steady all of these lines would be the same.

4.2 The Acceleration Field

Eulerian Description: Fixed pt.; different particles.

Lagrangian Description: Following individual particles. Newton's Second Law: $\vec{F} = m\vec{a}$

(a) **Material Derivative**

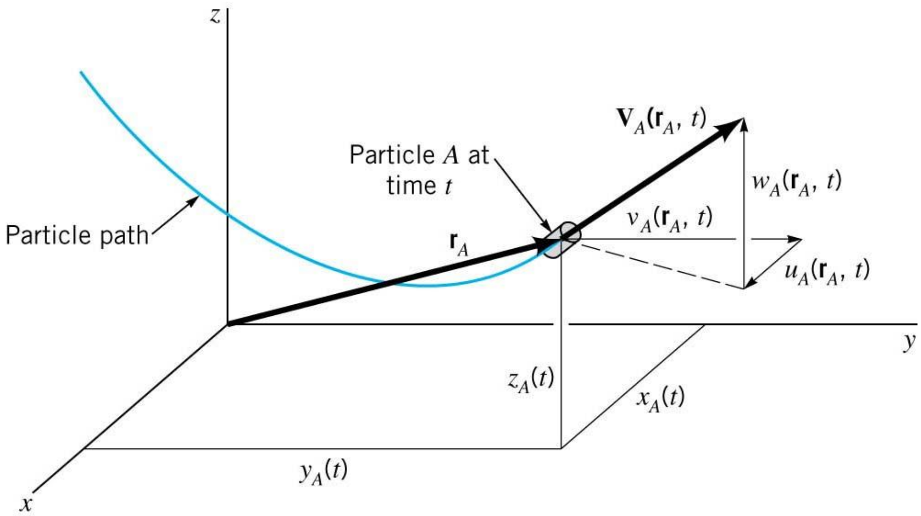


FIGURE 4.5 Velocity and position of particle *A* at time *t*.

$$\vec{V}_A = \vec{V}_A(r_A, t) = V_A(x_A(t), y_A(t), z_A(t), t)$$

$$\vec{a}_A = \frac{d\vec{V}_A}{dt} = \frac{\partial \vec{V}_A}{\partial t} + \frac{\partial \vec{V}_A}{\partial x}\frac{dx_A}{dt} + \frac{\partial \vec{V}_A}{\partial y}\frac{dy_A}{dt} + \frac{\partial \vec{V}_A}{\partial z}\frac{dz_A}{dt}$$ where

$$u = \frac{dx_A}{dt}; v = \frac{dy_A}{dt}; w = \frac{dz_A}{dt}$$

(chain rule of differentiation)

$$\therefore \underline{\vec{a} = \frac{D\vec{V}}{Dt} = \frac{\partial \vec{V}}{\partial t} + u\frac{\partial \vec{V}}{\partial x} + v\frac{\partial \vec{V}}{\partial y} + w\frac{\partial \vec{V}}{\partial z}}$$

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

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

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

$$\frac{D(\ )}{Dt} \equiv \frac{\partial(\ )}{\partial t} + u\frac{\partial(\ )}{\partial x} + v\frac{\partial(\ )}{\partial y} + w\frac{\partial(\ )}{\partial z}$$

$$= \frac{\partial()}{\partial t} + (\vec{V} \bullet \nabla)(\ )$$

Lagrangian description, Material, Substantial, Total derivative

(b) Unsteady and Convective Effects

$$\frac{D(\ )}{Dt} = \frac{\partial(\ )}{\partial t} + \left(\vec{V} \bullet \nabla\right)(\ )$$

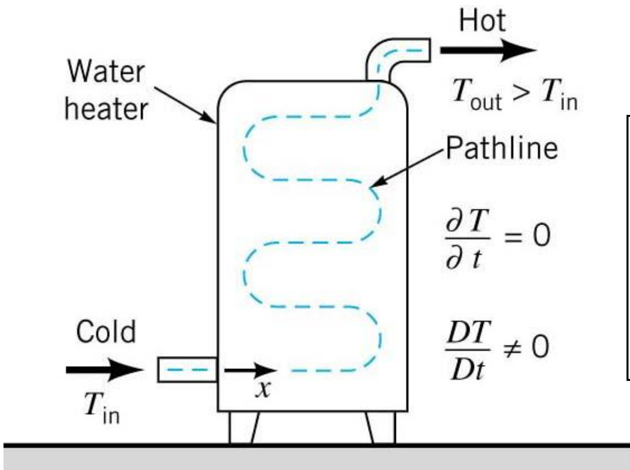
$\dfrac{\partial(\ )}{\partial t}$: local time derivative; unsteady effects

e.g. steady flows: $\dfrac{\partial \vec{V}}{\partial t}$ local acceleration = 0

e.g. unsteady cup of a coffee, where $\vec{V}$ = 0

Temperature variation: $\dfrac{\partial T}{\partial t} - (\vec{V} \bullet \nabla T) = \dfrac{\partial T}{\partial t} < 0$

$\left(\vec{V} \bullet \nabla\right)(\ )$: Convective derivative (acceleration: $(\vec{V} \bullet \nabla)\vec{V}$)



Steady state operation of a water heater

$$\frac{\partial \vec{V}}{\partial t} = \frac{\partial T}{\partial t} = 0$$

$$\frac{DT}{Dt} = \frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} = C$$

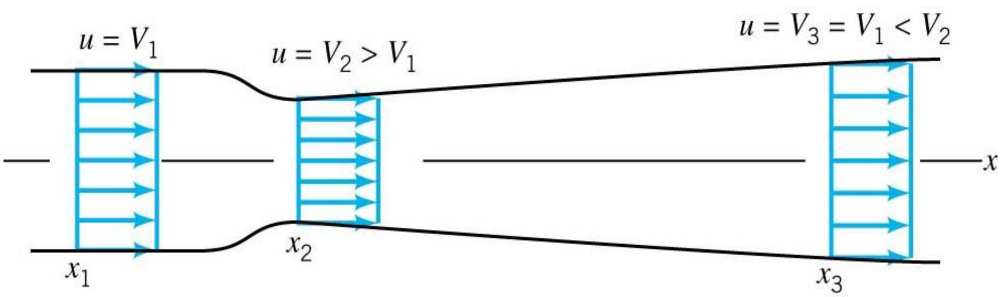


FIG. 4.8 Uniform, steady flow in a variable area pipe (1-D, steady flow

$$\frac{D\vec{V}}{Dt} = \frac{\partial \vec{V}}{\partial t} + (\vec{V} \bullet \nabla)\vec{V}$$

$$a_x = \frac{Du}{Dt} = \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} \neq 0$$

$$\frac{\partial u}{\partial t} = 0$$

x_1—x_2: $\frac{\partial u}{\partial x} > 0 \rightarrow a_x > 0$ (acceleration)

x_2—x_3: $\frac{\partial u}{\partial x} < 0 \rightarrow a_x < 0$ (deceleration)

(c) Streamline Coordinates:

A coordinate system defined in terms of the streamlines of the flow. In several flow situations, it is convenient to use streamline coordinates.

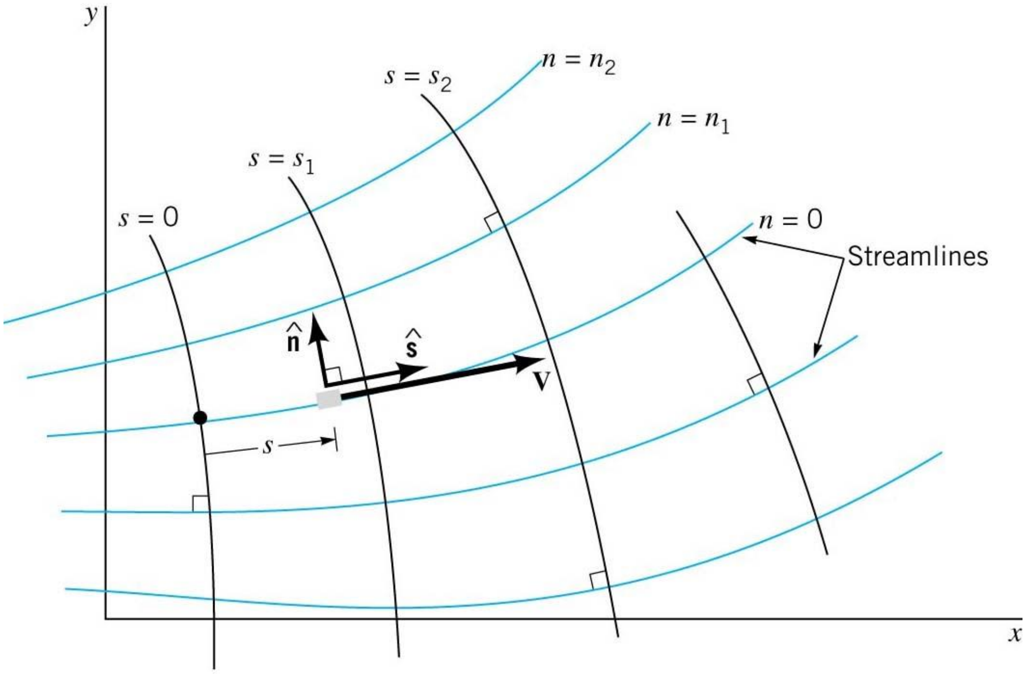


FIG. 4.9 Streamline coordinate system for 2-D flow.

$\vec{V} = V \cdot \vec{S}$; $\vec{a} = \dfrac{D\vec{V}}{Dt} = a_s\vec{S} + a_n\vec{n}$ along a streamline ($\vec{n}$ = constant)

By chain rule: $\vec{a} = \dfrac{D(V \cdot \vec{S})}{Dt} = \dfrac{DV}{Dt}\vec{S} + V\dfrac{D\vec{S}}{Dt}$

$$\vec{a} = \left(\frac{\partial V}{\partial t} + \frac{\partial V}{\partial S}\frac{dS}{dt} + \frac{\partial V}{\partial n}\frac{dn}{dt}\right)\vec{S} + V\left(\frac{\partial \vec{S}}{\partial t} + \frac{\partial \vec{S}}{\partial S}\frac{ds}{dt} + \frac{\partial \vec{S}}{\partial n}\frac{dn}{dt}\right)$$

$\because$ steady flow ($\dfrac{\partial}{\partial t} = 0$) & along a streamline ($\dfrac{dn}{dt} = 0 \; and \; \dfrac{dS}{dt} = V$)

$\therefore$ steady flow $\vec{a} = (V\dfrac{\partial V}{\partial S})\vec{S} + V(V\dfrac{\partial \vec{S}}{\partial S})$

where $\dfrac{\partial \vec{S}}{\partial S}$ is the change in the unit vector along the streamlines $|\vec{s}| = 1$

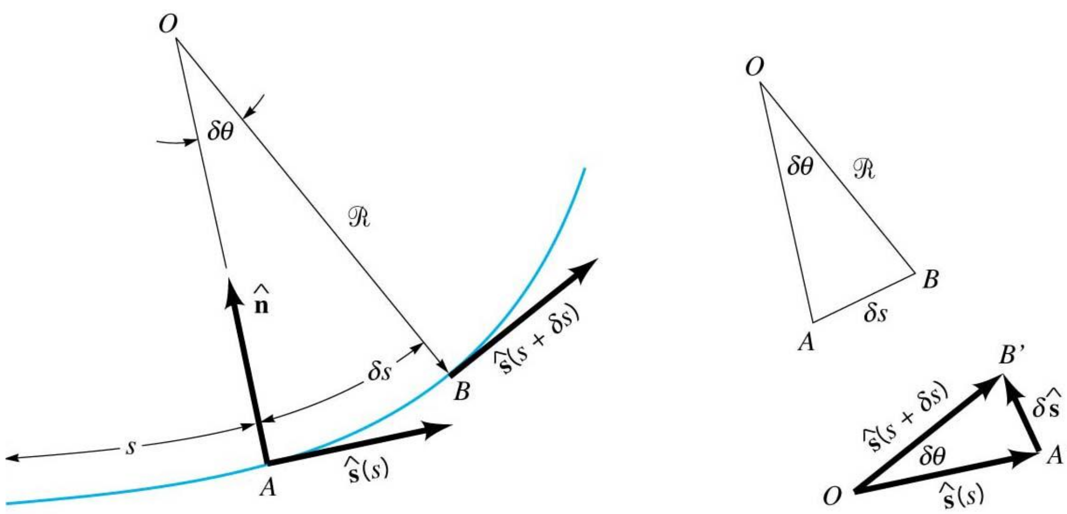


FIGURE 4.10 Relationship between the unit vector along the streamlines, $\vec{S}$, and the radius of curvature of the streamline, *R*.

$$\frac{\partial \vec{s}}{\partial s} = \lim_{\delta s \to 0} \frac{\delta \vec{s}}{\delta s} = \frac{\vec{n}}{R}$$

where *R* the radius of curvature of the streamline.

$$\because \vec{a} = V\frac{\partial V}{\partial s}\vec{s} + \frac{V^2}{R}\vec{n}$$

$a_s a_n$

convective acceleration centrifugal acceleration

4.3 Control Volume and System Representation

System: A specific, identifiable quantity of matter.

Control Volume: A volume in space through which fluid may flow.

Control Surface: surface of the control volume.

4.4 The Reynolds Transport Theorem

a. Extensive Property	Intensive Property	
B	b	(B=*m*b)
m (Mass)	1	
$\frac{1}{2}mv^2$ (K.E)	$\frac{v^2}{2}$	
mv (Momentum)	v	
etc	etc	

Reynolds Transport Theorem is an analytical tool for control volume and system representation.

$$B_{sys} = \lim_{\delta\forall \to 0} \sum_i \mathrm{b_i}(\rho_i \;\; \delta\forall_i) = \int_{sys} \rho b \, d\forall$$

$$\frac{dB_{sys}}{dt} = \frac{d(\int_{sys} \rho b \, \mathrm{d}\forall)}{\mathrm{dt}} \leftrightarrow \frac{dB_{c.v.}}{dt} = \frac{d(\int_{c.v.} \rho b \, \mathrm{d}\forall)}{\mathrm{dt}}$$

$\uparrow$

Reynolds Transport Theorem

b. Derivation of the R.T.T (See Fig. 4.11)

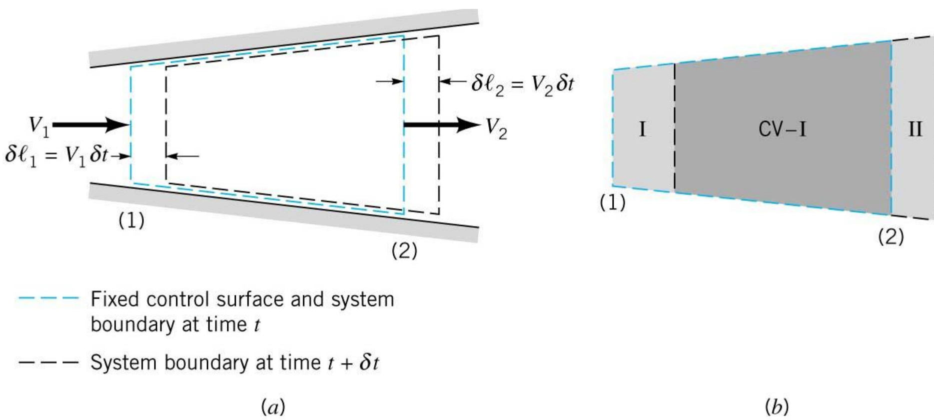


Fig. 4.11 Control volume and system for flow through a variable area pipe.

$B_{sys}(t) = B_{c.v.}(t)$

$$B_{sys}(t+\delta t) = B_{c.v.}(t+\delta t) - B_I(t+\delta t) + B_{II}(t+\delta t)$$

$$\lim_{\delta t \to 0} [\frac{\delta B_{sys}}{\delta t} = \frac{B_{sys}(t+\delta t) - B_{sys}(t)}{\delta t} = \frac{B_{c.v.} - B_I + B_{II} - B_{sys}}{\delta t}]$$

$$\frac{DB_{sys}}{Dt} = \lim_{\delta t \to 0} [\frac{B_{c.v.}(t+\delta t) - B_{c.v.}(t)}{\delta t} - \frac{B_I(t+\delta t)}{\delta t} + \frac{B_{II}(t+\delta t)}{\delta t}]$$

1 2

3

1. $$\frac{\partial B_{c.v.}}{\partial t} = \frac{\partial(\int_{c.v.} \rho b \, \mathrm{d}\forall)}{\partial \mathrm{t}};$$

2.

$$B_I(t+\delta t) = \rho_1 b_1(\delta\forall_I) = \rho_1 b_1(A_1 V_1 \delta t)$$

$$\overset{\bullet}{B}_{in} = \lim_{\delta t \to 0} \frac{B_I(t+\delta t)}{\delta t} = \rho_1 A_1 V_1 b_1$$

3. $$B_{II}(t+\delta t) = \rho_2 b_2(\delta\forall_{II}) = \rho_2 b_2(A_2 V_2 \, \delta t)$$

$$\overset{\bullet}{B}_{out} = \lim_{\delta t \to 0} \frac{B_{II}(t+\delta t)}{\delta t} = \rho_2 A_2 V_2 b_2$$

$$\frac{DB_{sys}}{Dt} = \frac{\partial B_{c.v.}}{\partial t} + \overset{\bullet}{B}_{out} - \overset{\bullet}{B}_{in}$$

$$= \frac{\partial B_{c.v.}}{\partial t} + \rho_2 A_2 V_2 b_2 - \rho_1 A_1 V_1 b_1$$

Simplified version of R.T.T. (4.15)

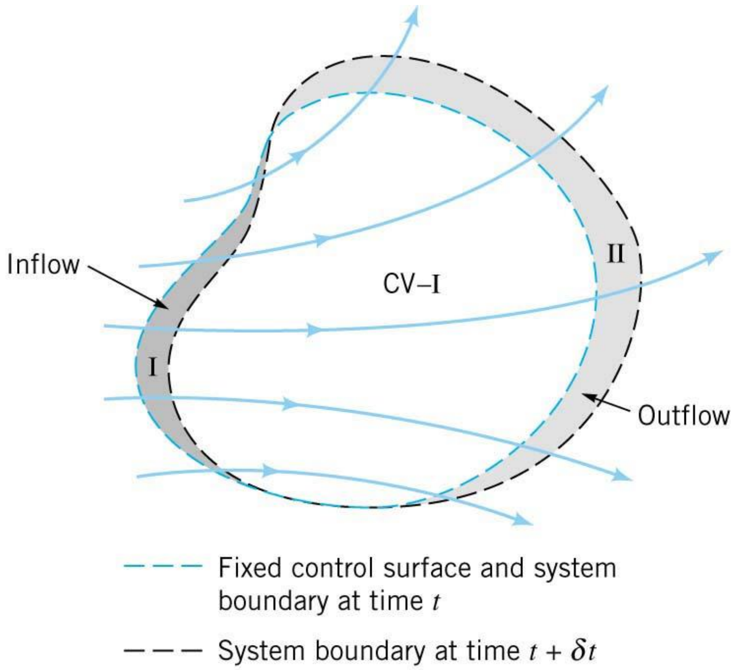


Fig. 4.13Control volume and system for flow through an arbitrary, fixed control volume.

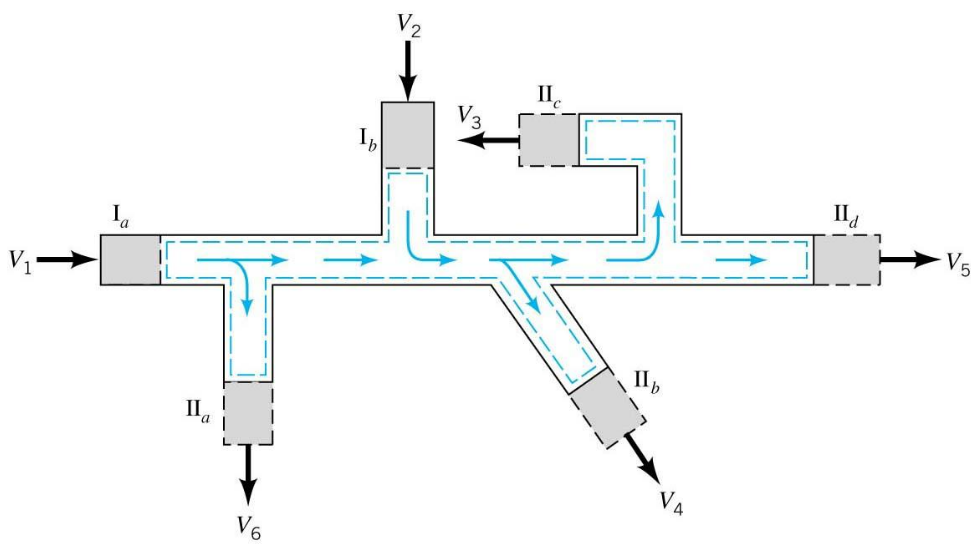


Fig. 4.13 Typical control volume with more than one inlet and outlet.

All inlets: I = Ia + Ib + Ic + …;

All outlets: II = IIa + IIb + IIc + …

Read Ex.4.8: $b = 1$; $B = m$; $A_1 = 0$

$$\frac{Dm_{sys}}{Dt} = \frac{\partial(\int_{c.v.} \rho \, d\forall)}{\partial t} + \rho_2 A_2 V_2$$

$\Downarrow$

0 (the amount of mass in a system is constant)

$$\frac{\partial(\int_{c.v.} \rho \, d\forall)}{\partial t} = -\rho_2 A_2 V_2$$

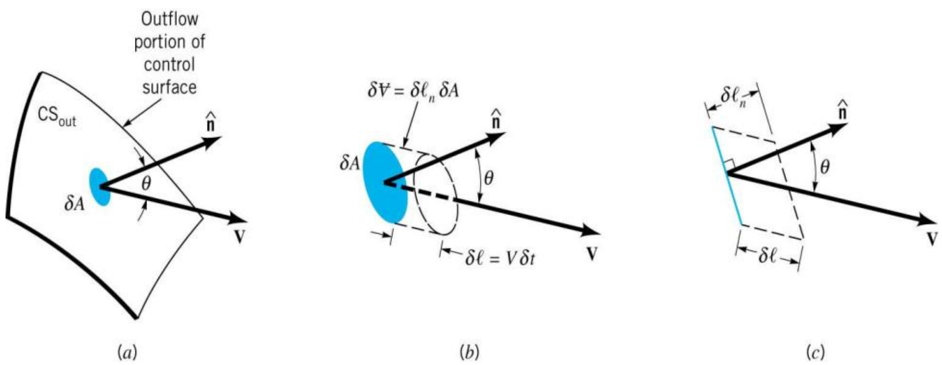


Fig. 4.14 Outflow across a typical portion of the control surface.

$$\delta \ell_n = \delta \ell \cos \theta$$

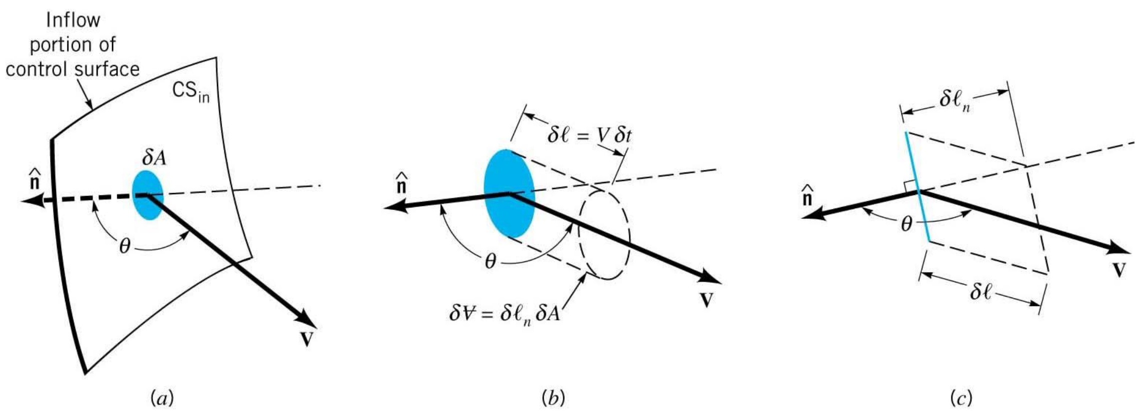


Fig. 4.16

Inflow across a typical portion ($\delta \ell_n = \delta \ell \cos \theta$)

$$\dot{B}_{out} = \int \delta \dot{B}_{out} = \int \lim_{\delta t \to 0} \frac{\rho b \, \delta\forall}{\delta t}$$

where $\delta\forall = \delta\ell_{\mathrm{n}} \delta A = \delta\ell \cos\theta \, \delta A = V\delta t \cos\theta \, \delta A$

$$\dot{\mathrm{B}}_{out} = \int \lim_{\delta t \to 0} \frac{\rho b v \cos\theta \, \delta t \cdot \delta A}{\delta t} = \int \rho b v \cos\theta \, \delta A$$

$$= \int_{\substack{c.s. \\ out}} d\,\dot{B}_{out} = \int_{\substack{c.s. \\ out}} \rho b v \cos\theta dA = \int_{\substack{c.s. \\ out}} \rho b \vec{v} \cdot \vec{n} \, dA$$

Similarly,

$$\dot{B}_{in} = -\int_{\substack{cs \\ in}} \rho b v \cos\theta \, dA = -\int_{\substack{cs \\ in}} \rho b \vec{v} \cdot \vec{n} \, dA$$

$$\dot{B}_{out} - \dot{B}_{in}$$

$$= \int_{\substack{cs \\ out}} \rho b \vec{v} \cdot \vec{n} \, dA - \int_{\substack{cs \\ in}} \rho b \vec{v} \cdot \vec{n} \, dA = \int_{cs} \rho b \vec{v} \cdot \vec{n} \, dA$$

$$\therefore \frac{DB_{sys}}{Dt} = \frac{\partial B_{c.v}}{\partial t} + \int_{c.s} \rho b \vec{v} \cdot \vec{n} \, dA$$

$$= \frac{\partial}{\partial t} \int_{cv} \rho b \, \mathrm{d}\forall + \int_{\mathrm{cs}} \rho b \vec{v} \cdot \vec{n} \, dA \qquad (4.19)$$

For a fixed, nondeforming C.V.

B = mb; b = 1 (mass); b = v (momemtun); b = $0.5v^2$ (energy)

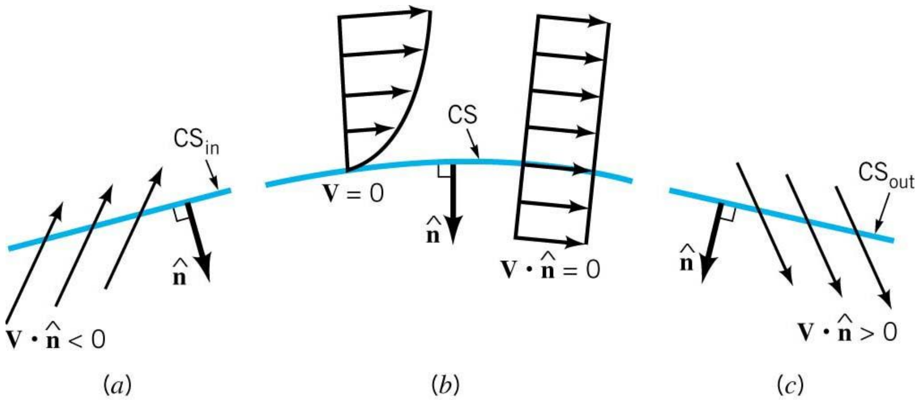


Fig. 16Possible velocity configurations on portions of the control surface: (a) inflow, (b) no flow across the surface, (c) outflow.

c. Relationship to Material Derivative.

$$\frac{D(\ )}{\mathrm{Dt}} = \frac{\partial(\ )}{\partial \mathrm{t}} + u\frac{\partial(\ )}{\partial \mathrm{x}} + v\frac{\partial(\ )}{\partial y} + w\frac{\partial(\ )}{\partial \mathrm{z}}$$

Following a fluid particle or system. The material derivative is essentially the infinitesimal (or derivative) equivalent of the finite size (or integral) Reynolds transport theorem.

d. Steady effects: $\frac{\partial(\ )}{\partial t} = 0$; $\frac{DB_{sys}}{Dt} = \int_{cs} \rho b \vec{v} \cdot \vec{n} dA$

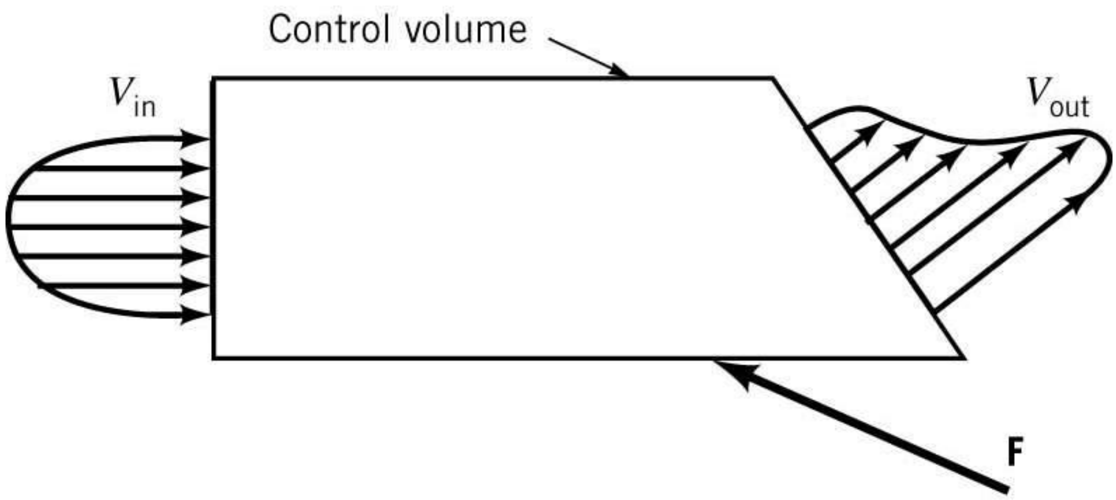


Fig. 4.17 Steady flow through a control volume

Unsteady effects: $\frac{\partial(\ )}{\partial t} \neq 0$; $\frac{DB_{sys}}{Dt} = \frac{\partial}{\partial t} \int_{cv} \rho b d\forall$

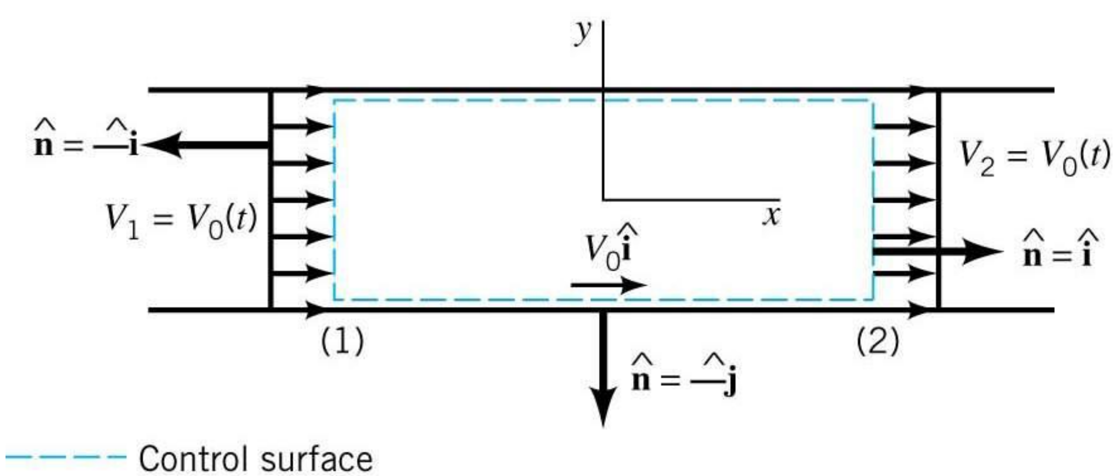


Fig. 4.18Unsteady flow through a constant diameter pipe.

$$\int_{CS} \rho b \vec{v} \cdot \vec{n} dA = 0$$

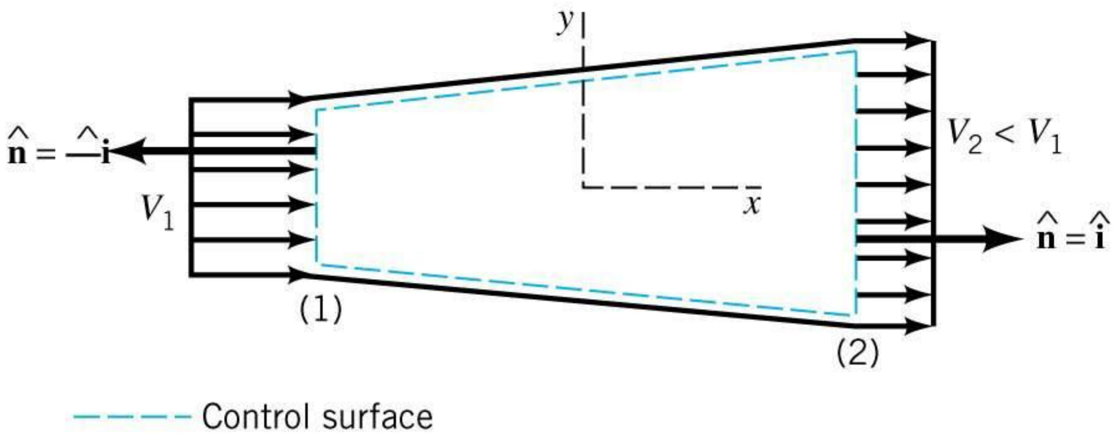


Fig. 4.20

Flow through a variable area pipe

$$\int_{CS} \rho b \vec{v} \cdot \vec{n} dA \neq 0$$

e. Moving Control Volume

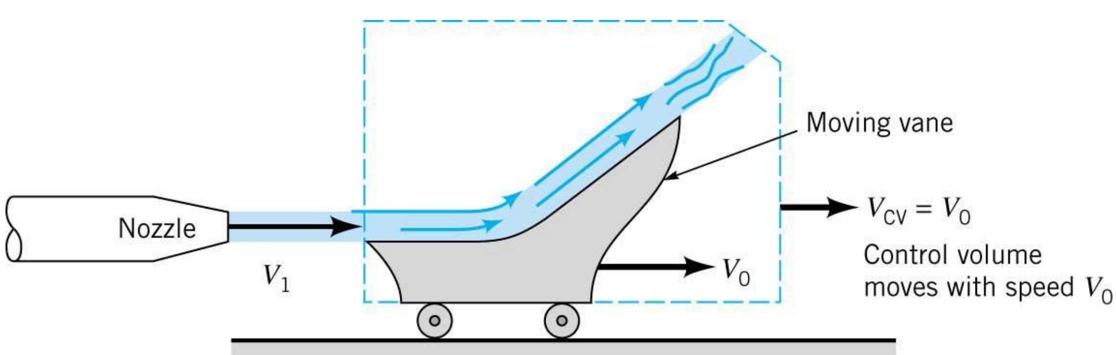


Fig. 4.21 Example of a moving control volume

$V_{c.v.} = V_0$; *C.V.*可以等速或加速變形

（僅考慮*C.V.*以等速變形）<u>***V*r= *W*= *V*abs- *V*c.v.**</u>

$$\frac{DB_{sys}}{Dt} = \frac{\partial}{\partial t}\int_{c.v.} \rho b \, \mathrm{d}\forall + \int_{\mathrm{c.s.}} \rho b \vec{v}_r \cdot \vec{n} \, dA$$

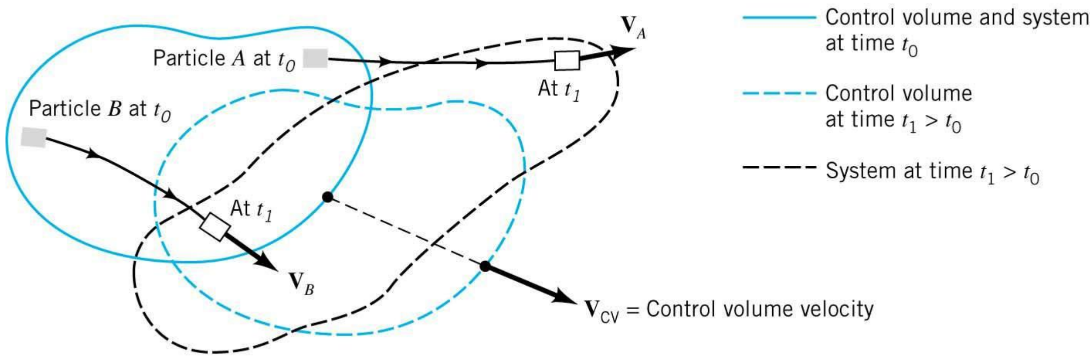


Fig. 4.21 Typical moving control volume and system

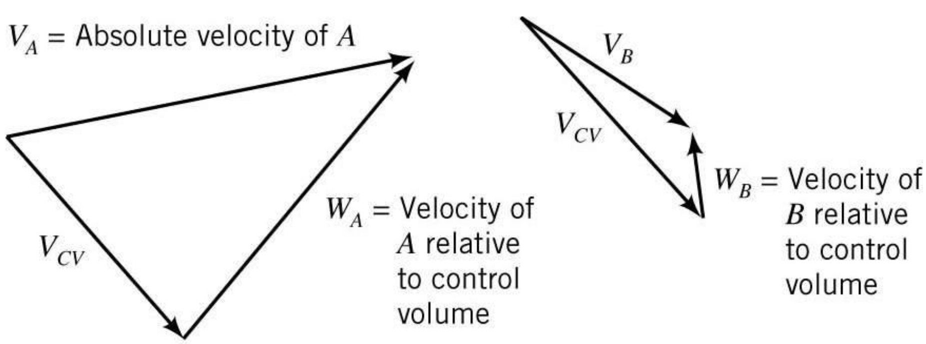


Fig. 4.22 Relationship between absolute and relative velocities

f. Selection of a *C.V.*

Any volume in space can be considered as a *C.V.*. The selection of an appropriate *C.V.* in fluid mech. is very similar to the selection of an appropriate free-body diagram in static or dynamics.

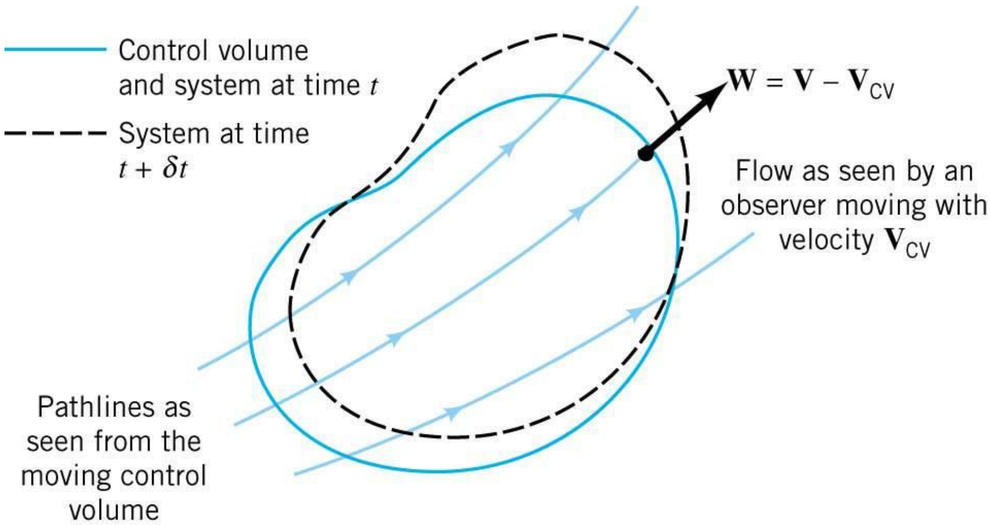


Fig. 4.24 Control volume and system as seen by an observer moving with the control volume

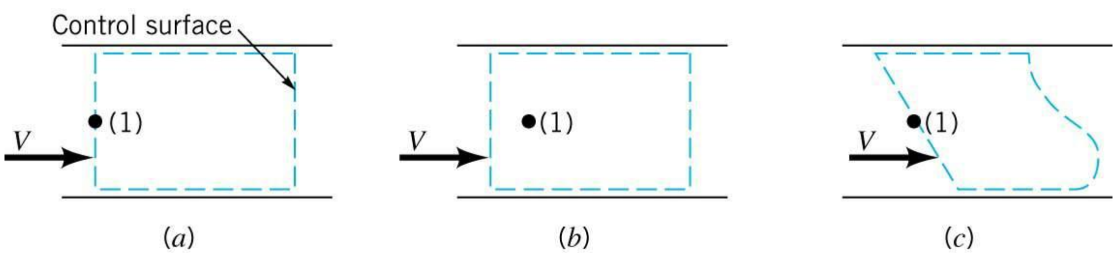


Fig. 4.25

Various control volumes for flow through a pipe.

Key Words and Topics

Problems 4.60 and 4.61

Review Ch 1 ~ Ch 4

Potential Function (ϕ):

- Definition: $u = \frac{\partial \phi}{\partial x}$ and $v = \frac{\partial \phi}{\partial y}$

- Characteristic: It always satisfies the irrotationality (i.e., $\zeta_z = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0$)

- Physical meaning: ϕ = constant is a potential line

  Streamline and potential line are orthogonal to each other

Potential Flow:

- Governing equation: $\nabla^2 \phi = 0$ or $\nabla^2 \psi = 0$

To Solve Potential Flow Problems:

- Superposition of Elementary Flows
    - Basic elementary flows:
        - Uniform flow
        - Free vortex
        - Source/Sink
        - Doublet

- Method of Image

Superposition:

For example: Flow over a circular cylinder = Uniform flow + Doublet

Uniform flow: $\psi_{uniform} = U\left(y\cos\alpha - x\sin\alpha\right)$

Doublet: $\psi_{doublet} = -\dfrac{\lambda\sin\theta}{r}$

Flow over a circular cylinder: $\psi = \psi_{uniform} + \psi_{doublet}$

Method of Image:

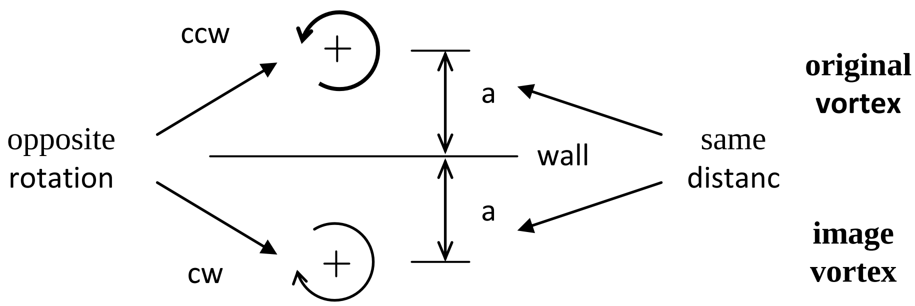
- Used to simulate ground effects

Solution Procedure:

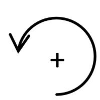
Step 1: Draw image vortices so that resultant velocity normal to wall is zero

Image vortices are constructed as:

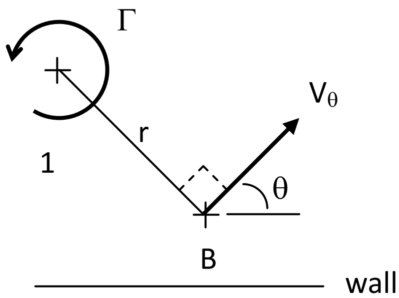
- Same distance below the wall
- Opposite rotation



Step 2: Find induced velocity at location B (point of interested) by all vortices (original + images)

- $v_\theta = \dfrac{K}{r} = \dfrac{\Gamma}{2\pi r}$, $\Gamma > 0$ if 

For example, $\vec{V}_B$ induced by vortex 1 (original vortex):



	Magnitude	Sign
x – comp:	$V_\theta \cos\theta$ (+)	
y – comp:	$V_\theta \sin\theta$	(+)

$$\vec{V}_{B,1} = (V_\theta \cos\theta \,, V_\theta \sin\theta)$$

Step 3: Find stream function ψ at any location (x, y)

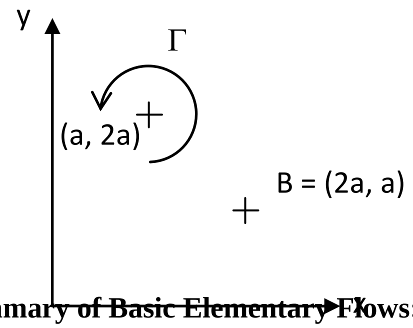
- $\psi = -\dfrac{\Gamma}{2\pi}\ln r = -\dfrac{\Gamma}{4\pi}\ln r^2$ where Γ> 0 is in the counter-clockwise direction

<u>Example:</u>

A positive line vortex with strength □ is located at a distance (x, y) = (a, 2a) from the corner.

1) Compute the total induced velocity at point B, where (x, y) = (2a, a).

2) Find the stream function ψ at any location (x, y).



Summary of Basic Elementary Flows:

	Stream Function ψ	Potential Function ϕ	**Velocity Components**
Uniform flow at angle α with the x axis (Eq. 8.14)	$\psi = U\left(y\cos\alpha - x\sin\alpha\right)$	$\phi = U\left(x\cos\alpha + y\sin\alpha\right)$	$u = U\cos\alpha$ $v = U\sin\alpha$

Source or Sink (Eq. 4.131) m>0: Source m<0: Sink	$\psi = m\theta$	$\phi = m \ln r$	$v_r = \frac{m}{r}$ $v_\theta = 0$
Vortex (Eq. 4.132) Γ>0: ccw motion Γ<0: cw motion	$\psi = -K \ln r$ $K = \frac{\Gamma}{2\pi}$ (Eq. 8.16)	$\phi = K\theta$	$v_r = 0$ $v_\theta = \frac{K}{r}$
Doublet (Eq. 8.28)	$\psi = -\frac{\lambda \sin\theta}{r}$	$\phi = \frac{\lambda \cos\theta}{r}$	$v_r = -\frac{\lambda \cos\theta}{r^2}$ $v_\theta = -\frac{\lambda \sin\theta}{r^2}$

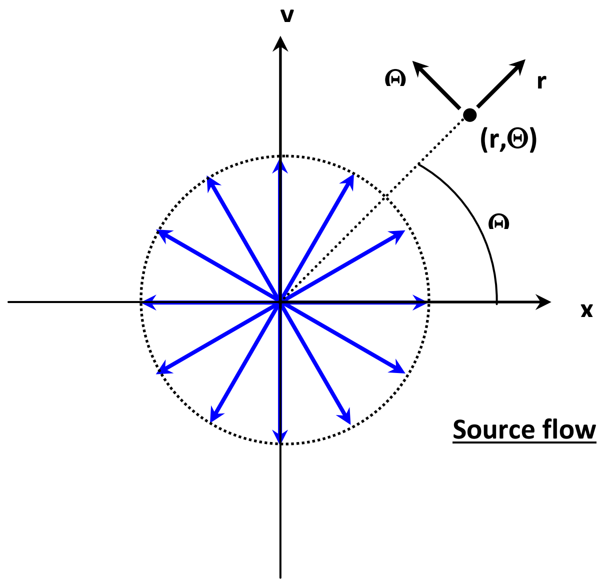
Velocity components are related to the stream function and potential function through the relationships:

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

Elementary Singularities

We explore some special flows now, which satisfy the Laplace equation, but are physically unrealistic. The interesting fact about these is, although they are singular in nature, they can provide physically meaningful flows when they are combined with other flows. We will use the flows mostly in cylindrical coordinates.

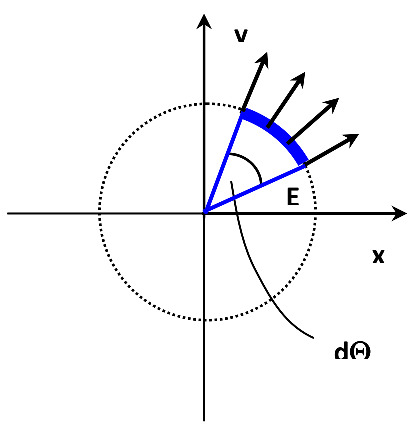
Source Flow: **A source flow is defined at a point as the flow that creates new fluid particles continuously. In 2-dimensions a source located at the origin will create fluid streamlines as shown below:**



Since the streamlines are all radial, the source flow velocity components may be written as $V_r \neq 0$, $V_\Theta = 0$. **We define the strength of a source, q, as the volumetric flow per unit depth through any closed circuit enclosing the source.** **Volumetric flux by definition is** $Q = \int_A \vec{V} \bullet d\vec{A}$ **.**

Let us choose the circuit as a circle of radius, E.

Let $W = \text{Depth}$



$$\therefore d\vec{A} = (E \bullet d\Theta) \bullet W\hat{e}_r$$

$$\therefore \vec{V} = V_r\hat{e}_r + V_\Theta\hat{e}_\Theta$$

$$\vec{V} \bullet d\vec{A} = (W\,E\,d\Theta)V_r$$

Or, $$Q = \int_A \vec{V} \bullet d\vec{A} = WE\int_{\Theta=0}^{2\pi} V_r\, d\Theta$$

$$\therefore q = \frac{Q}{W} = E\int_{\Theta=0}^{2\pi} V_r\, d\Theta$$

Because of the symmetry about the origin, V_r will not be a function of Θ. Thus, $q = V_r E(2\pi)$ **or,** $V_r = \dfrac{q}{(2\pi)E}$ **through the circle of radius E. In general,** $V_r = \dfrac{q}{(2\pi)r}$ **through a circle of radius r.**
Note that the appearance of V_r indicates the flow has an infinite V_r as $r \rightarrow \infty$**. Thus, a source flow is considered as singular at the origin. We may define a sink flow in the same manner as a "negative source" (q < 0).**

Let us find the stream function for a source (or, sink) flow.

$$\therefore V_r = \frac{q}{2\pi r} = \frac{1}{r}\frac{\partial \psi}{\partial \Theta} \text{ and } V_\Theta = -\frac{\partial \psi}{\partial r} = 0$$

It's easy to show by integration:

$$\psi_{\substack{source \\ (or,\text{sink})}} = \frac{q\Theta}{2\pi}$$

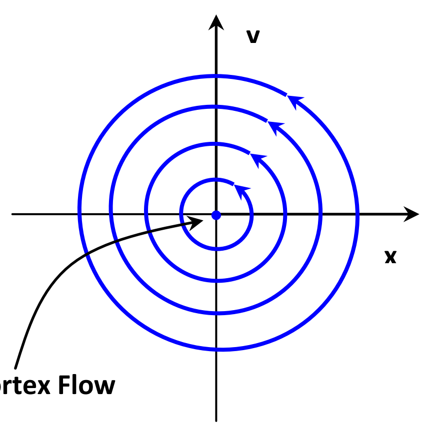
Note that we have omitted the constant of integration in the formula above. First, if the constant is dropped it does not change the velocity field at all (velocity components involve derivatives of ψ).

Moreover, to plot streamlines, we must set the $\psi = \text{constant} = \frac{q\Theta}{2\pi}$, and select different values of the constant.

Henceforth the remaining singularity functions will be presented without additional constants in their representation.

Vortex Flow: **A counterclockwise vortex located at the origin has circular streamlines as shown.**

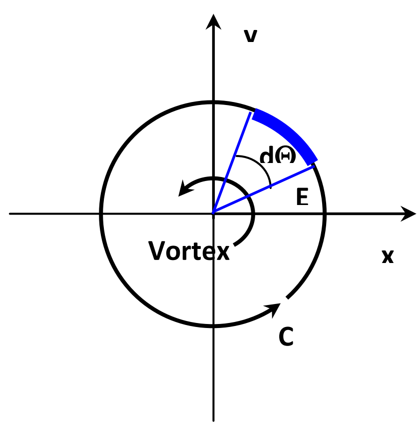
In cylindrical coordinates, this means $V_r \neq 0$, $V_\Theta = 0$. Furthermore, we define the strength of the vortex, Γ, as the circulation around any closed curve enclosing the vortex.



As before, let the closed curve be a circle of radius E.

Since circulation $= \oint_C \vec{V} \bullet d\vec{r}$ **:**

$$d\vec{r} = E\, d\Theta\, \hat{e}_\Theta$$



$$\vec{V} \bullet d\vec{r} = V_\Theta \; E\, d\Theta$$

$$\therefore \Gamma = \oint_C \vec{V} \bullet d\vec{r} = V_\Theta \; E \int_0^{2\pi} d\Theta$$

$= E\, V_{\Theta}(2\pi)$

or, $V_{\Theta} = \dfrac{\Gamma}{2\pi E}$ **around circle of radius, E.**

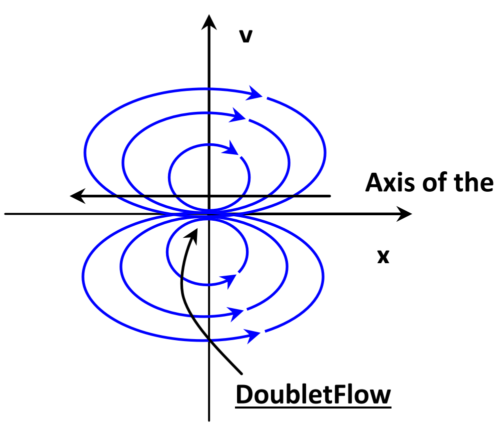
[$\because$ V_{Θ} is not a function of Θ, by symmetry]

In general, $V_{\Theta} = \dfrac{\Gamma}{2\pi E}$**,** $V_r = 0$ **for a counterclockwise vortex located at the origin. It yields a stream function given by:**

$$\psi_{\text{vortex}} = -\frac{\Gamma}{2\pi}\ln r$$

Similarly, a clockwise vortex will give $\psi = \dfrac{\Gamma}{2\pi}\ln r$ **. Again, these are singular at the origin (as** $r \to 0$ **).**

Doublet Flow:A doublet (or, dipole) is like an electric magnet. It produces a streamline pattern same as what you have seen in physics by spreading "iron dust" on a piece of paper with a magnet underneath.



A doublet is obtained by bringing a source and a sink close together. Assume that a source and sink of strength "q" and"-q" are placed at a distance "l" apart. As the two singularities are brought closer to each other (i.e., l $\to$ **0), suppose we hold** $ql = \mu =$ **constant. Then we will create the flow field given by the streamline pattern to the left. A doublet (unlike source or vortex) has no symmetry at the origin. Also it has an axis as shown (directed from the sink to the source inside the doublet). The relevant velocity component and stream function representation is given below:**

$$V_r = \frac{\mu \cos \Theta}{2\pi r^2} \qquad V_\Theta = \frac{\mu \sin \Theta}{2\pi r^2} \qquad \Psi_{doublet} = \frac{\mu \sin \Theta}{2\pi r}$$

The above formulae hold for a doublet axis along the positive x-axis.

Continue

Sources, Sinks and Doublets - the Building Blocks of Potential Flow

In the previous handout we developed the following equation for the velocity potential:

$$\boxed{\begin{array}{l}\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = 0 \\ Or \\ \nabla^2 \phi = 0\end{array}} \qquad (1)$$

where the operator ∇^2 is called the Laplacian operator. This equation holds for 2-D and 3-D inviscid irrotational flows. If we are only interested in 2-D irrotational inviscid flows, we may also solve for:

$$\boxed{\nabla^2 \psi = 0} \qquad (2)$$

where ψ is the stream function.

After we have solved for the velocity potential or the stream function, we can compute the velocities. In a Cartesian coordinate system, for 2-D flows, we will use:

$$\boxed{\begin{array}{l}u = \frac{\partial \phi}{\partial x} = \frac{\partial \psi}{\partial y} \\ v = \frac{\partial \phi}{\partial y} = -\frac{\partial \psi}{\partial x}\end{array}}$$

(3)

In a polar coordinate system, for 2-D flows we will use:

$$\boxed{\begin{aligned} v_r &= \text{Radial velocity} = \frac{\partial \phi}{\partial r} = \frac{1}{r}\frac{\partial \psi}{\partial \theta} \\ v_\theta &= \text{Tangential velocity} = \frac{1}{\mathrm{r}}\frac{\partial \phi}{\partial \theta} = -\frac{\partial \psi}{\partial r} \end{aligned}}$$

(4)

In 3-D, the velocities are given only in terms of the velocity potential, as follows:

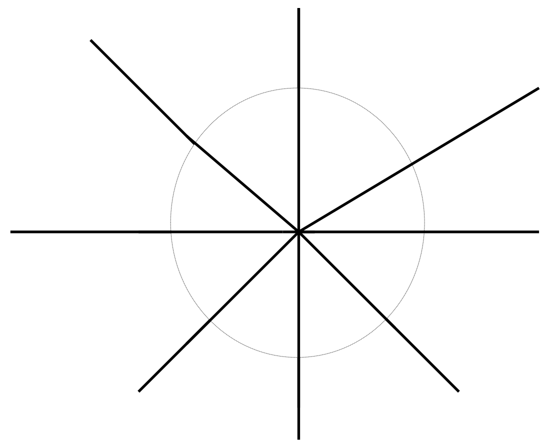
$$\boxed{\begin{aligned} &\vec{V} = \vec{\nabla}\phi \\ &Or, \\ &u = \frac{\partial \phi}{\partial x} \qquad v = \frac{\partial \phi}{\partial y} \qquad w = \frac{\partial \phi}{\partial z} \end{aligned}}$$

(5)

Once the velocity is known, we can find pressure from the Bernoulli's equation.

In this section, we consider some simple solutions to the Laplace's equation (1 or 2). Since equation 91) and 92) are linear, we can superpose many such simple solutions to arrive at a more complex flow field. This is like building a complex configuration using Lego blocks. The individual simple solutions are the individual Lego pieces, which on their own, are not very interesting. Together, however, they can solve some very interesting flows, including flow over airfoils and wings.

Building Block #1: 2-D Sources and Sinks: A source is like a lawn sprinkler. It sprays the water (or air) radially, and equally, in all the directions, at the rate of Q units per unit time. If this is a sink (e.g. a drain hole on a concrete pavement) the velocity vectors will still be radial, but directed inwards towards the center. The sign of Q will be positive for a source, and negative for a sink.



Consider a circle of radius r enclosing this source. Let v_r be the radial component of velocity associated with this source (or sink). Then, form conservation of mass, for a cylinder of radius r, and unit height perpendicular to the paper,

$$
\begin{aligned}
&Q = (2\pi r)\cdot(1)\cdot v_r \\
&Or, \\
&v_r = \frac{Q}{2\pi r}
\end{aligned}
\tag{6}
$$

Solving equation (6) for the velocity potential and the stream function we get, for a source or a sink:

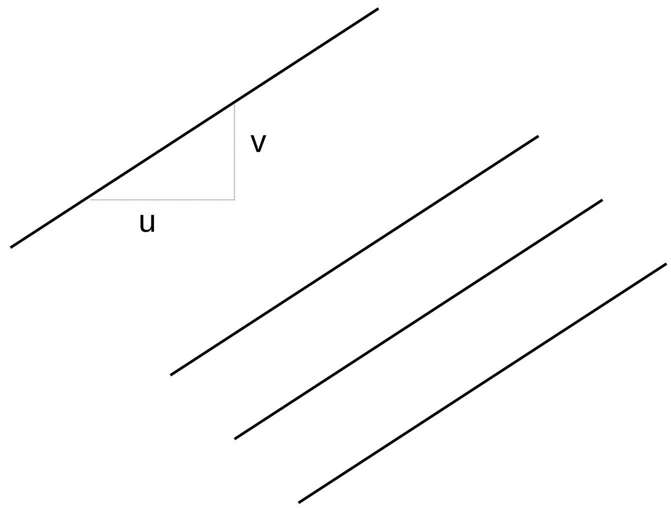
$$
\boxed{\begin{aligned}
\phi &= \frac{Q}{2\pi}\log_e r \\
\psi &= \frac{Q}{2\pi}\theta
\end{aligned}}
\tag{7}
$$

plus a constant. The constants appear as we integrate the velocity to get the velocity potential or stream function. Every simple solution we consider will be an analytical function (like equation 7) plus a constant to be determined later.

<u>Exercise</u>: Verify for yourselves that (7) satisfies Laplace's equation in polar coordinates:

$$\nabla^2\phi = \frac{\partial^2\phi}{\partial r^2} + \frac{1}{r}\frac{\partial\phi}{\partial r} + \frac{1}{r^2}\frac{\partial^2\phi}{\partial\theta^2} = 0$$
$$\nabla^2\phi = \frac{\partial^2\psi}{\partial r^2} + \frac{1}{r}\frac{\partial\psi}{\partial r} + \frac{1}{r^2}\frac{\partial^2\psi}{\partial\theta^2} = 0$$

Building Block #2: Uniform Flow:



This is also, by itself, an uninteresting flow. It represents a uniform flow with the velocity components u_∞ and v_∞ along the x- and y- axes. The stream function and the velocity potential associated with this flow are:

$$\boxed{\begin{aligned}\phi_{\text{Uniform Flow}} &= u_\infty x + v_\infty y \\ \psi_{\text{Uniform Flow}} &= u_\infty y - v_\infty x\end{aligned}} \tag{8}$$

If we use equation (3) on the definitions given in equation (8), we recover the Cartesian components of velocity. Notice that these functions shown on (8) are simple straight lines. It is also easy to see that these functions given in equation (8) satisfy the Laplace's equation.

Superposition of a Source and a Uniform Flow:

Let us try to superpose the uniform flow and the flow field due to a source. All we have to do is add the flow field given in equations (8) to equation (9). The result is given below:

$$\boxed{\begin{aligned}&\phi = \frac{Q}{2\pi}\log_e r + u_\infty x + v_\infty y\\ &\psi_{\text{Uniform Flow}} = u_\infty y - v_\infty x + \frac{Q}{2\pi}\theta\end{aligned}}$$

(10)

We can take x and y- derivatives of this flow field to get the velocity field. Note that the quantity 'r' represents the distance between where the source (x_s,y_s), and a general point (x,y) where the distance is being computed. That is,

$$r = \sqrt{(x - x_{Source})^2 + (y - y_{Source})^2}$$

(11)

Thus, the velocity potential in Cartesian form, taking into account where the source has been placed, is:

$$\phi = \frac{Q}{4\pi}\log\left[(x - x_s)^2 + (y - y_s)^2\right] + u_\infty x + v_\infty y$$

(12)

and the velocities are:

$$\begin{aligned}u &= \frac{\partial\phi}{\partial x} = \frac{Q}{8\pi}\frac{(x - x_s)}{(x - x_s)^2 + (y - y_s)^2} + u_\infty\\ v &= \frac{\partial\phi}{\partial y} = \frac{Q}{8\pi}\frac{(y - y_s)}{(x - x_s)^2 + (y - y_s)^2} + v_\infty\end{aligned}$$

(13)

These velocities may be plugged into the Bernoulli's equation:

$$p + \frac{1}{2}\left(u^2 + v^2\right) = p_\infty + \frac{1}{2}\left(u_\infty^2 + v_\infty^2\right)$$

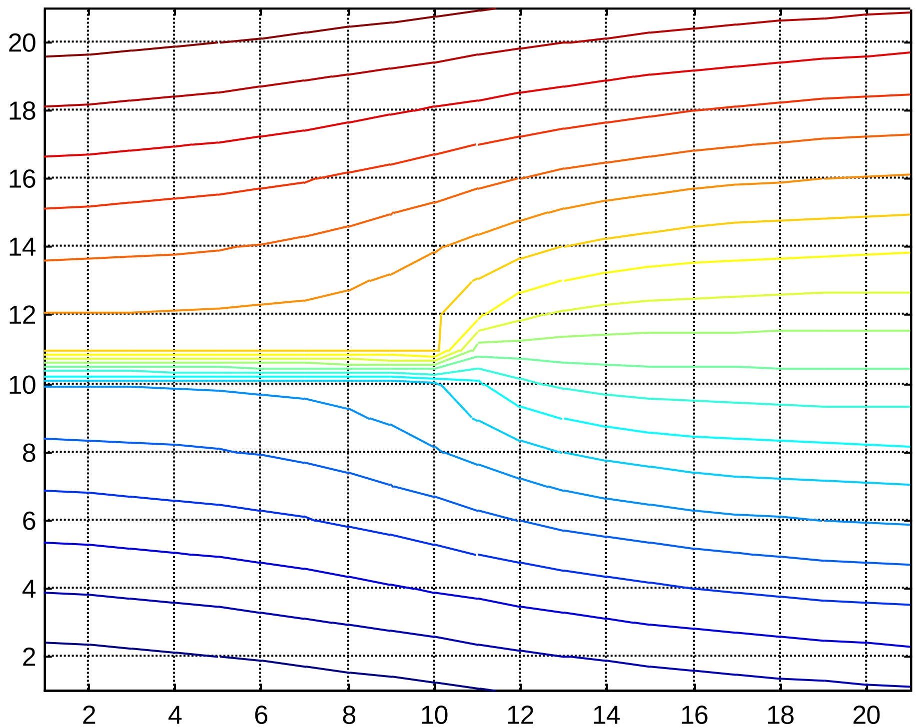
(14)

where p_∞ is the pressure value far away from the source.

We can also plot the flow field and the streamlines. The easiest way to accomplish this is using built-in functions such as the MATLAB function "contour". Here is an example. In this example, a source of strength Q equal to unity is placed at the origin. The freestream velcoity is UINF= 1, and VINF=0. Here is the MATLAB script for modeling this flow.

```
X=-1:.1:1;

Y=X;

    [x,y] = meshgrid(X,Y);


 Q=1;

 UINF=1.;

 VINF=0.;

 z=Q/(2.*3.14158)*atan2(y,x)+UINF*y-VINF*x;

 contour(z,20);
```

This example produces the contours shown below. Note that this looks like flow around the nose of a body. This body is called "Rankine's Half-Body."



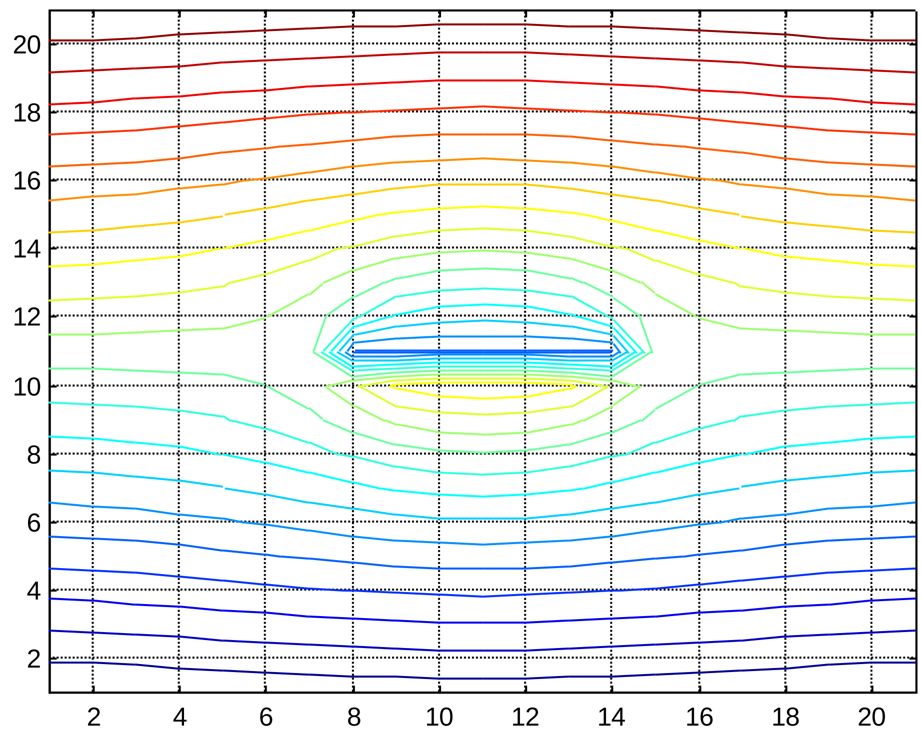
Superposition of Uniform flow, source and a sink:

We can superpose a source placed at (X1,Y1), a sink of equal strength placed at (X2,Y2), and a uniform velocity. Let us say, for the sake of illustration, that the source is placed at x=-0.3, and the sink is placed at x= +0.3. on the x- axis. Let us assume that Q=1, UINF=1 and VINF=0.0. Then we can look at the contours by executing the following MATLAB script:

```
%Contours of  Stream function caused by a source of strength Q placed in a

%         uniform stream with Cartesian components u=UINF and v=VINF.

% The source is placed in this example at X= -0.3, and the sink is

% placed at x= +.3


X=-1:.1:1;

Y=X;
```

```
[x,y] = meshgrid(X,Y);

Q=1;
UINF=1.;
VINF=0.;
z=Q/(2.*3.14158)*atan2(y,x+.3)-Q/(2.*3.14158)*atan2(y,x-.3)+UINF*y-VINF*x;
contour(z,20);
```

Here are the contours from the resulting plot. This flow resembles flow over an oval shaped object, called “Rankine’s full body”. It is similar to the shape made popular in Ford commercials.



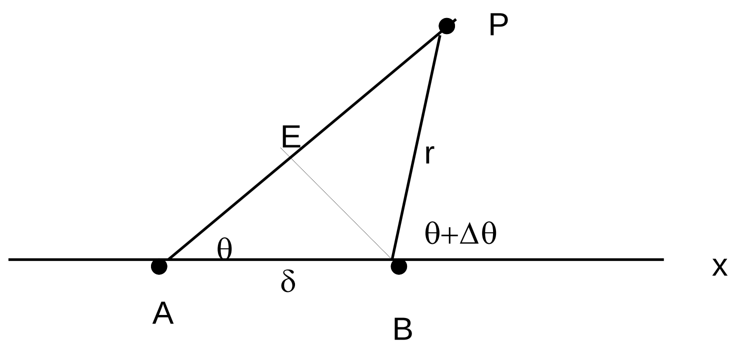
<u>Doublets:</u>

Doublets are source-sink pairs, initially separated by a distance □ , which are brought close together by making the separation distance □ →0. To keep them from annihilating each other, their strength Q is progressively increased so that Q times □ remains a constant. This constant is given the symbol □ , called the strength of the doublet.

We can derive expressions for the stream function (and the velocity potential) for a doublet from the known expressions for sources and sinks. Consider a source of strength Q placed on the x-axis at a point A, and a sink of strength –Q placed on the x- axis at point B. The points A and B are placed a distance δ apart. Then, the stream function at a general point P in the flow field is given by:

$$\psi_{Doublet} = \underset{\delta \to 0}{Limit} \frac{Q}{2\pi}\left(\theta_1 - \theta_2\right)$$

where θ_1 is the angle formed by the line AP with respect to the x- axis, and θ_2 is the angle formed by the line BP with respect to the x- axis. See the figure below.



In the figure above, the angle EPB is $(\theta_2 - \theta_1) = \Delta\theta$.

The distance BP ≈ The distance EP = r. Then, for small values of $\Delta\theta$, EB ≈ $r\sin(\Delta\theta) \approx r\Delta\theta$

Consider next the right angle triangle ABE. For this triangle, EB/AB = $\sin\theta$. Using

EB = $r\Delta\theta$, and AB = δ we get

$$\Delta\theta = \delta \sin\theta / r$$

Thus, stream function associated with the doublet, in the limit as δ goes to zero, is given by:

$$\boxed{\psi_{Doublet=} = -\frac{\mu}{2\pi}\frac{\sin\theta}{r}}$$

If we superpose the doublet, and a uniform flow we get:

$$\psi = u_\infty y - v_\infty x - \frac{\mu}{2\pi}\frac{\sin\theta}{r}$$

If the uniform flow is parallel to the x- axis, using y=rsin□ □ and defining a^2=□ □ □ □ V_∞ we get:

$$\psi = u_\infty y\left(1 - \frac{a^2}{r^2}\right)$$

Notice that the stream function ψ is zero on the surface r=a. In other words, r=a, a cylinder is a streamline. Thus, the superposition of a doublet and a uniform flow, for some special situations, becomes flow over a circular cylinder. We can plot this function using MATLAB. We will find that this function does yield flow over a circular cylinder of radius a.

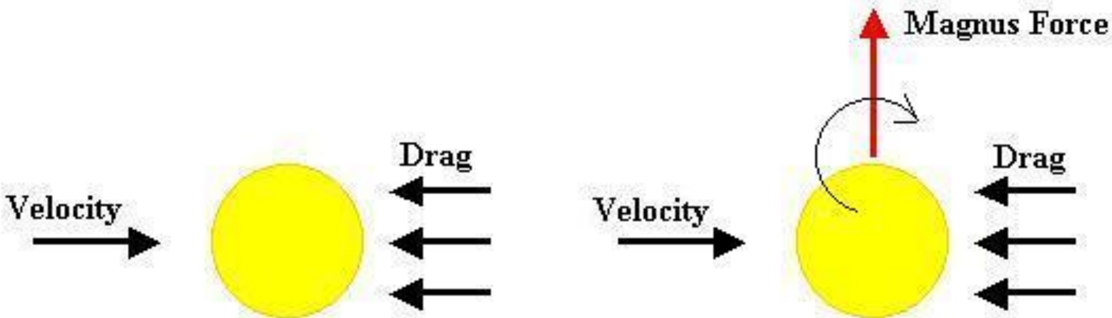
Magnus Effect

What you need

- Two polystyrene cups.
- Sticky tape.
- Two large rubber bands.

What you do

1. Use sticky tape to fix the bottoms of the polystyrene cups together.
2. Knot the rubber bands together.
3. Hold the rubber band in the centre of the cups and wrap the bands around about twice. Finish with the end of the elastic bands on the bottom pointing away from you.
4. Hold the cup in one hand and the end of the elastic in your other hand.
5. Pull back the cups and let go.
6. With enough practice you should be able to make the flying cups loop in the air.



Ping-pong ball without spin
(a)

Ping-pong ball with spin
(b)

What's going on?

This is known as the Magnus effect, and it is the reason why top footballers can make balls curve in the air and how golfers can make golf balls perform some amazing aerodynamics.

The cups are fired forward because of the stretched elastic band. If we ignore the fact the cups are spinning we can see that air will flow over the cups from front to back in a fairly uniform way.

However, in this system, when the cups are released the bands unwind and the cups are forced to spin. If the bands are wound correctly the cups will be given back spin; the bottom of the cups move forwards while the top is moving backwards. Because of the rough surface of the cups, air is trapped near the surface and moves with the cups as they spin.

The top of the cups has air moving from front to back as they spins, and the cups also have air flowing over them from front to back because they are flying through the air. The bottom of the cups also have air moving from the front to the back because they are flying through the air, but, crucially, the bottom also has air moving back to the front because of the direction of the spinning cups. Therefore, the cups are sitting in air which is moving very differently at different parts: there is fast moving air at the top while the air is close to being stationary at the bottom.

Faster air has a lower pressure, so the cups have low pressure above them and higher pressure underneath. The cups are forced upwards.

As improbable as it seems, it is possible to make the cups travel backwards. To understand how you have to realise that the force making the cups lift is at right angles to the cups' forward motion. As the cups starts to rise vertically they also experience a force at right angles to their new 'forward' motion. This lift force actually makes the cup move back towards you. On this return part of the loop the flow at the top and bottom of the cups are reversed, the cup is forced down, and then eventually forward along its original path.

The air resistance which allows the layer of air to stick to the surface of the cups also slows the cups down. It slowly stops the cups from spinning and as the spin is reduced so the lift vanishes. The cups start to drop and eventually hit the floor.

Solutions of Boundary Layer Equations

Now we develop the solution strategies for the boundary layer equations given by Prandtl. Remember in his 2 equations of continuity and x-momentum, **the pressure gradient term is assumed to be known.**

Continuity: $$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad \text{(A)}$$

x-momentum: $$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{dp}{dx}+\nu\frac{\partial^2 u}{\partial y^2} \qquad \text{(B)}$$

Thus this set becomes mathematically solvable. There are two approaches to solve boundary layer equations. We shall present both here. However the emphasis will be in the second approach since it is easier to work with and gives an insight to the behavior of fluid particles in the boundary layer. The standard approaches are:

(i) Exact solution method (Blasius' Solution)
(ii) Approximate Solution Method (Karman-Pohlhausen Method)

The second approach is also called the momentum integral method. We begin with the exact solution method given by Blasius.

Exact Solution Method

Blasius performed a **transformation technique** to change the set of two partial differential equations (A and B) into a **single ordinary differential equation**. He solved the boundary layer over a flat plate in external flows. If we assume the plate is oriented along the x-axis, we may neglect the pressure gradient term, i.e., $\frac{\partial p}{\partial x}=0$. The traditional approach before Blasius was to drop out the continuity equation from the set by the introduction of the stream function $\Psi(x,y)$. With this definition:

$$\text{(B)} \Rightarrow \frac{\partial \Psi}{\partial y}\frac{\partial^2 \Psi}{\partial x \partial y}+\left(-\frac{\partial \Psi}{\partial x}\right)\left(\frac{\partial^2 \Psi}{\partial y^2}\right)=\nu\frac{\partial^3 \Psi}{\partial y^3}$$

However, Blasius used this equation in non-dimensional variables. Let us define $\eta(x,y)$ as a single variable by: $\boxed{\eta=\frac{y}{\sqrt{\frac{\nu x}{U}}}}$ or $y\sqrt{\frac{U}{\nu x}}$. It is easy to verify that η will be non-dimensional by substituting the units of ν, U, χ and y. He also introduced the non-dimensional stream function given by: (Correction: Please replace χ by x in the section below)

$$f(\eta)=\frac{\Psi}{\sqrt{\nu\chi U}}$$

Using mathematical manipulation from calculus, we may write:

$$u=\frac{\partial \Psi}{\partial y}=\frac{\partial \Psi}{\partial \eta}\frac{\partial \eta}{\partial y}=\sqrt{\nu\chi U}\cdot f'(\eta)\cdot\sqrt{\frac{U}{\nu\chi}}=U\cdot f'(\eta)$$

$$\therefore\ f'(\eta)=\frac{u}{U} = \text{non-dimensional velocity function}$$

$$v=-\frac{\partial \Psi}{\partial x}=-\left[\frac{1}{2}\sqrt{\frac{\nu U}{\chi}}\cdot f(\eta)+\sqrt{\nu\chi U}\cdot\frac{\partial f}{\partial x}\right] \qquad \text{(by chain rule)}$$

$$= -\left[\frac{1}{2}\sqrt{\frac{\nu U}{\chi}}\cdot f(\eta) + \sqrt{\nu\chi U}\cdot\frac{\partial f}{\partial \eta}\frac{\partial \eta}{\partial x}\right]$$

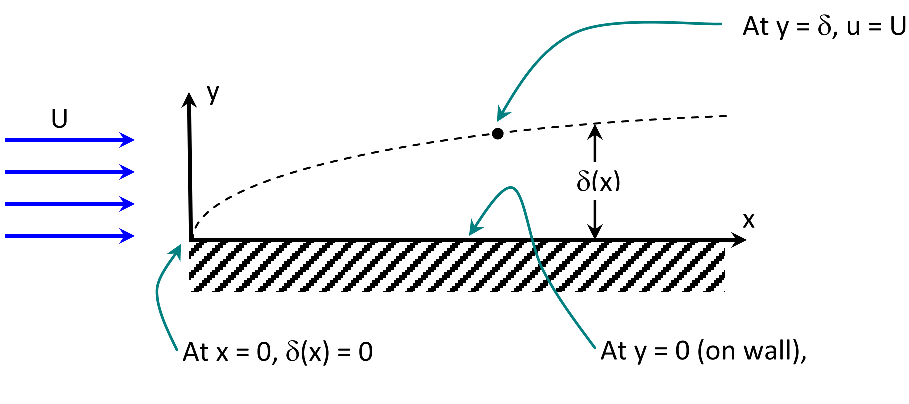
But since $\frac{\partial \eta}{\partial x} = y\sqrt{\frac{U}{\nu}}\left(-\frac{1}{2}x^{-\frac{3}{2}}\right) = -\frac{\eta}{2x}$, we may simplify v into $v = \frac{1}{2}\sqrt{\frac{\nu U}{\chi}}\left[\eta f' - f\right]$.

Similarly we can show:

$$\frac{\partial u}{\partial x} = -\frac{1}{2}\frac{\eta U}{\chi}\cdot f'', \qquad \frac{\partial u}{\partial y} = U\sqrt{\frac{U}{\nu\chi}}\cdot f''$$

and $$\frac{\partial^2 u}{\partial y^2} = \left(\frac{U^2}{\nu\chi}\right)\cdot f'''$$

Therefore, the original x-momentum can be written as $\boxed{2f''' + f\cdot f'' = 0}$ upon simplifications. Note that this is an ordinary differential equation with η as the independent variable and f is the dependent variable. To solve this third order equation we need three boundary conditions. Let us check the figure below.



We may write at y = 0, u = v = 0 into a different form: $\boxed{\text{At } \eta = 0,\ \ f = f' = 0}$. Similarly as y →∞, u = U may be written as

$$\boxed{At\ \eta \to \infty,\ \ f' = 1}$$

Using these three boundary conditions the solution of the governing equation may be obtained by the use of power series solution and shown in the table below. The important things to note are the points corresponding to the edge of the boundary layer. Since u ≈ U, $f' \approx 1$, (we choose the value of .9915, since δ is defined at u = .99U). Thus $\eta = y\sqrt{\frac{U}{\nu\chi}} = 5.0$ from the table below:

$\eta = y\sqrt{\frac{U}{\nu x}}$	f	$f' = \frac{u}{U}$	f''
0	0	0	0.3321
0.5	0.0415	0.1659	0.3309
1.0	0.1656	0.3298	0.3230
1.5	0.3701	0.4868	0.3026
2.0	0.6500	0.6298	0.2668
2.5	0.9963	0.7513	0.2174
3.0	1.3968	0.8460	0.1614
3.5	1.8377	0.9130	0.1078
4.0	2.3057	0.9555	0.0642
4.5	2.7901	0.9795	0.0340
5.0	3.2833	0.9915	0.0159
5.5	3.7806	0.9969	0.0066
6.0	4.2796	0.9990	0.0024
6.5	4.7793	0.9997	0.0008
7.0	5.2792	0.9999	0.0002
7.5	5.7792	1.0000	0.0001
8.0	6.2792	1.0000	0.0000

Using the alternate definition, $\eta = \frac{y}{\delta}$, we get

$$\delta \cong \frac{5.0}{\sqrt{\frac{U}{\nu\chi}}} \quad \text{or,} \quad \frac{\delta}{\chi} = \frac{5.0}{\sqrt{Re_x}}$$

Now, $$\tau_{yx} = \mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \quad where\,\frac{\partial v}{\partial x} \to 0 \ (\because v << u)$$

Therefore, the wall shear stress, τ_w, may be written as

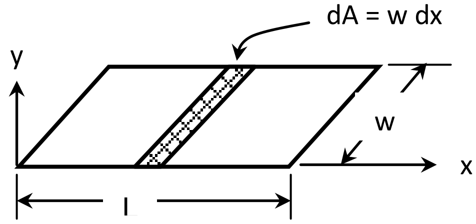
$$\tau_w = \mu\left.\frac{\partial u}{\partial y}\right|_{y=0} = \frac{.332\rho U^2}{\sqrt{Re_x}}$$

We define **Skin Friction Coefficient** as the non-dimensional wall shear stress, given by:

$$C_f \triangleq \frac{\tau_w}{\frac{1}{2}\rho U^2} = \frac{0.664}{\sqrt{Re_x}}$$

In this case both $\delta(x) = \frac{5.0\chi}{\sqrt{Re_x}}$ and $C_f = \frac{0.664}{\sqrt{Re_x}}$ are claimed to be exact solution of steady, laminar boundary layer over a flat plate oriented along the x-axis.

We notice from the above expressions that both $\delta(x)$ and C_f change along the plate. While $\delta(x)$ increases (boundary layer grows) with χ, $C_f \to 0$ as $x \to \infty$. Both quantities depend on the variable Reynolds number, $Re_x\left[= \frac{U\chi}{\nu}\right]$. If the plate length is not infinite, how do we obtain the shear force on it? We may do this by integrating directly or, the use of the concept of "overall Skin Friction Coefficient". For example, for a finite length, L, of the plate, the shear force $F_{yx} = \int_A \tau_{yx}\big|_{y=0} dA$



where, $dA = w dx$.

but: $$\tau_{yx}\big|_{y=0} = \tau_w = C_f\left(\frac{1}{2}\rho U^2\right)$$

Therefore the $C_f(x) = \dfrac{0.664}{\sqrt{Re_x}}$ may be substituted above and F_{yx} obtained by integration.

Alternately, define $\overline{C}_f$ = Overall Skin Effect Coefficient $\underline{\Delta} \dfrac{1}{L}\int_0^L C_f(x)\,dx$. Thus the $\overline{C}_f$ is nothing but "length-averaged" friction coefficient. Unlike $C_f(x)$, $\overline{C}_f$ is a constant value for the whole plate.

Similarly the average shear stress for the plate may be defined as $\overline{\tau}_w = \dfrac{1}{L}\int_0^L \tau_w(x)\,dx$. Finally, the shear force on the plate may be written as the product of $\overline{\tau}_w$ and the plate area.

Approximate Solution Method

Unlike the Blasius solution, which is exact, approximate solution **method assumes an approximate shape of the velocity profile.** This velocity profile is then utilized to evaluate quantities related to the governing differential equation, given below by Karman and Pohlhausen. This method, which is called the momentum integral method, changes the two equations given by Prandtl into a single differential equation. This equation over a flat plate may be written as:

$$\boxed{\overline{\tau}_w = \rho U^2 \frac{d\Theta}{dx}}$$

where, τ_x =Wall shear stress
ρ = Density of fluid
U = Free stream velocity
and, Θ = Momentum thickness of the boundary layer

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

The above equation is applicable only when the pressure gradient term is zero. For the case of **non-zero pressure gradients** you should use

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

Velocity Profiles

Since the Karman-Pohlhausen method **requires an assumed velocity profile**, let us explore some velocity profiles and their characteristics (see example problem 1). For example, suppose we assume the velocity profile to be a second order polynomial $u(y) = A + By + Cy^2$ where A, B, and C are constants.

To evaluate velocity profile constants A, B, and C, we must use boundary conditions. The following three conditions may be used:

1) No-slip: $y = 0$, $u = 0$
2) B.L. Edge Velocity: $y = \delta$, $u = U$
3) B.L. Edge Shear: $y = \delta$, $\frac{du}{dy} = 0$

Note that at the edge of the definededgeoftheboundarylayer $u = .99U$ and $\frac{du}{dy} \approx 0$. However we approximate them with the rounded values. This is the reason the solution method by Momentum Integral Method is considered an approximate one.

With the above profile,

1) $\Rightarrow \quad 0 = A + B\cdot(0) + C\cdot(0)^2 \Rightarrow A = 0$

2) $\Rightarrow \quad U = A + B\cdot(\delta) + C\cdot(\delta)^2 \Rightarrow \frac{U}{\delta} = B + C\cdot\delta \qquad [\because A = 0]$

3) $\Rightarrow \left.\frac{du}{dy}\right|_{y=\delta} = B + 2C\delta = 0$

Subtracting the second condition from the third,

$$C\delta = -\frac{U}{\delta} \Rightarrow C = -\frac{U}{\delta^2}$$

Using this in the second condition, $\frac{U}{\delta} = B - \frac{U}{\delta} \Rightarrow B = \frac{2U}{\delta}$

$$\therefore u(y) = \frac{2U}{\delta} y - \frac{U}{\delta^2} y^2$$

$$\text{or, } \boxed{\frac{u}{U}(y) = 2\left(\frac{y}{\delta}\right) - \left(\frac{y}{\delta}\right)^2} \Rightarrow \text{Parabolic Profile (see plot in the example)}$$

Remember the use of the boundary layer velocity profile is only meaningful when $0 \le y \le \delta$. The use of this velocity profile may now be made to obtain Θ and τ_w

$$\Theta = \int_0^\delta \frac{u}{U}\left(1 - \frac{u}{U}\right) dy = \delta \int_0^1 \frac{u}{U}\left(1 - \frac{u}{U}\right) d\left(\frac{y}{\delta}\right)$$

$$\text{or,} \qquad \frac{\Theta}{\delta} = \int_0^1 \left[2\frac{y}{\delta} - \left(\frac{y}{\delta}\right)^2\right]\left[1 - 2\frac{y}{\delta} + \left(\frac{y}{\delta}\right)^2\right] d\frac{y}{\delta}$$

Note that defining a new variable $\eta = \dfrac{y}{\delta}$ makes the evaluation much easier.

$$\frac{\Theta}{\delta} = \int_0^1 \left(2\eta - \eta^2\right)\left(1 - 2\eta + \eta^2\right) d\eta$$

$$= \int_0^1 \left(2\eta - \eta^2 - 4\eta^2 + 2\eta^3 + 2\eta^3 - \eta^4\right) d\eta$$

$$= \int_0^1 \left(2\eta - 5\eta^2 + 4\eta^3 - \eta^4\right) d\eta$$

$$= 1 - \frac{5}{3} + 1 - \frac{1}{5} = \frac{15 - 25 + 15 - 3}{15} = \frac{2}{15}$$

Similiarly, $\tau_w = \mu \left.\dfrac{du}{dy}\right|_{y=0}$ $[\because v \approx 0$ in boundary layer]

$$= \frac{\mu U}{\delta} \bullet \left.\frac{\partial(u/U)}{\partial(y/\delta)}\right|_{y=0} = \frac{\mu U}{\delta} \bullet \left.\frac{\partial(u/U)}{\partial\eta}\right|_{\eta=0}$$

$$= \frac{\mu U}{\delta} \bullet \left[2 - 2\eta\right]_{\eta=0} = \frac{2\mu U}{\delta} \text{ for the parabolic profile}$$

Using the above results for Θ and τ_w in the momentum integral equation for a flat plate gives

$$\frac{2\mu U}{\delta} = \rho U^2 \bullet \left(\frac{2}{15}\right)\frac{d\delta}{dx}$$

Separating the variables δ and x, and integrating

$$\int_{\delta=0}^{\delta(x)} \delta\, d\delta = \int_{x=0}^{x} 15\left(\frac{\mu}{\rho U}\right)dx$$

$$\frac{\delta^2}{2} = 15\left(\frac{\mu}{\rho U}\right) \Rightarrow \boxed{\delta^2 = \frac{30\mu x}{\rho U}}$$

To express the boxed equation in a non-dimensional form divide both sides by x^2,

$\left(\frac{\delta}{x}\right)^2 = \frac{30\mu}{\rho U x} = \frac{30}{Re_x}$, where $Re_x = \frac{\rho U x}{\mu}$ is the Reynolds number based upon the variable x.

$$\therefore \frac{\delta}{x} = \sqrt{\frac{30}{Re_x}} = \frac{5.48}{\sqrt{Re_x}}$$

Compare this result with the earlier exact solution obtained under the Blasius method.

$$\therefore \frac{\delta}{x} = \frac{5.0}{\sqrt{Re_x}}$$

We therefore see the popularity of the parabolic velocity profile. Although the solution by Karman-Pohlhausen method is approximate it gives less than 10% error when compared with the exact solution is laminar flows over a flat plate.

Now that we have obtained δ(x), the shear stress, $\tau_{w,}$ and skin friction coefficient, C_f, may be obtained for the parabolic profile.

$$\tau_w = \frac{2\mu U}{\delta} = \frac{2\mu U\sqrt{Re_x}}{5.48x} = .365\frac{\mu U}{x}\sqrt{Re_x}$$

$$\therefore C_f(x) = \frac{\tau_w}{\frac{1}{2}\rho U^2} = \frac{.73}{\rho U^2} \bullet \frac{\mu U}{x} \bullet \sqrt{Re_x} = \frac{.73}{\sqrt{Re_x}}$$

This is comparable with $C_f(x) = \frac{.664}{\sqrt{Re_x}}$ obtained earlier in the exact solution method.

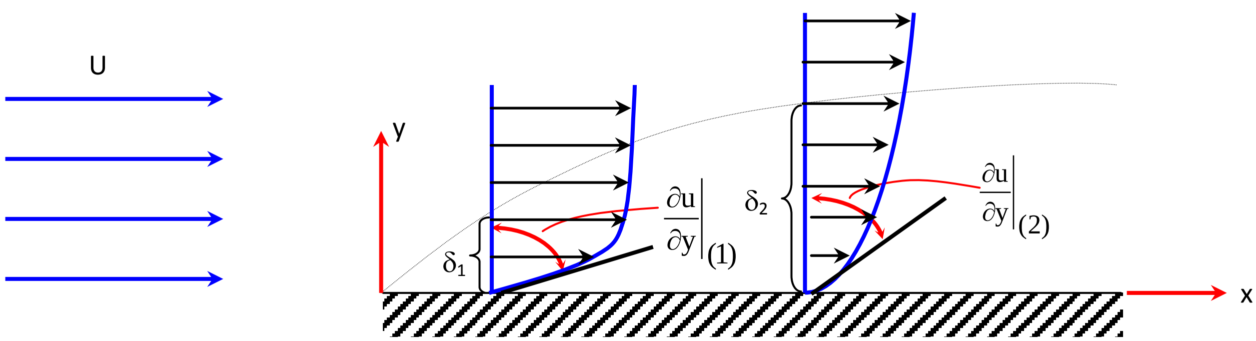
To summarize, we have obtained the growth of the boundary layer δ(x) and the skin friction characteristic $C_f(x)$ as a solution of the boundary layer equations by the exact and approximate methods. Once $C_f(x)$ is known, the shear stress and skin friction force may be evaluated (see examples).

As stated before, the **frictional forces are not the dominant forces in high-speed flows**. The component of drag due to skin friction is called the friction drag. Thus friction drag is significantly lower than pressure drag in boundary layers of high Reynolds number flows. However**, Prandtl found a very**

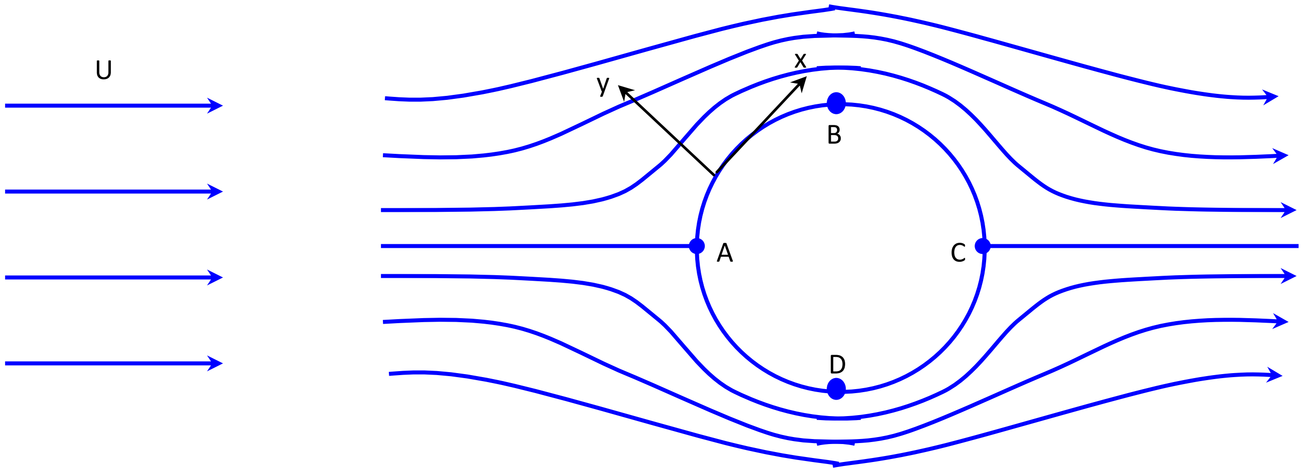
important influence of these small frictional forces in controlling the pressure drag. To understand this we must investigate the phenomenon of flow separation.

Flow Separation and Boundary Layer Control

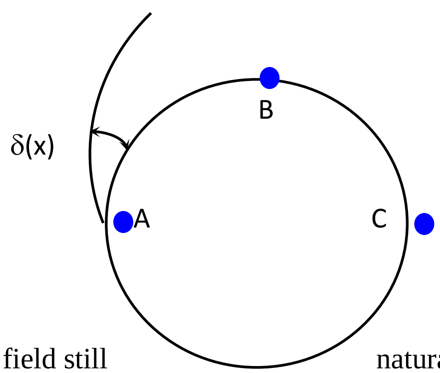
Earlier we noted that as the boundary layer over a flat plate grows, the value of the skin friction coefficient goes down. This may be explained from the fact that as more fluid layers are decelerated due to shear at the plate shear values near the plate need to be as large compared to the entrance region of the plate.



Compare the station (2) ($\delta = \delta_2$) with station (1) ($\delta = \delta_1$). The shear on the plate at (2) is smaller since the shear angle $\left.\frac{\partial u}{\partial y}\right|_{(2)} < \left.\frac{\partial u}{\partial y}\right|_{(1)}$. Mathematically, we know $C_f(x) \to 0$ as $Re_x \to \infty$. But can the shear go to zero on the flat plate, and if so, what are the physical implications? The answer depends on the physical configurations. For a flat plate, shear may never go to zero as $C_f \to 0$ only when $Re_x \to \infty$ or $x \to \infty$. However if we get some assistance from the pressure gradient, C_f can be zero much earlier. Consider, for this purpose, flow over a circular cylinder.

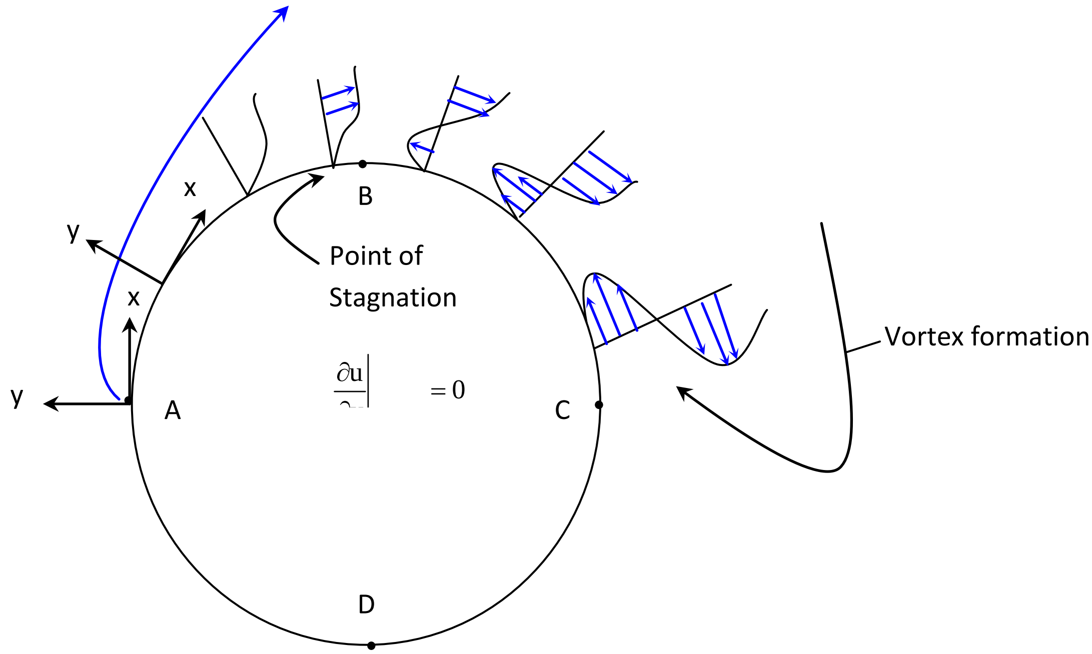


The figure above shows a circular cylinder in steady, ideal flow, U. The stagnation points are A and C, while the maximum velocity points are B and D. Since the regions A to B and A to D accelerate the flow, $\frac{dp}{dx} < 0$ (note x is in the tangential direction along the cylinder). Similiarly the regions B to C and D to C are the adverse pressure gradient regions ($\frac{dp}{dx} < 0$). Now imagine if this cylinder was placed in a real flow, viscous boundary layer will start to grow from the front stagnation point A, slowing the fluid particles.



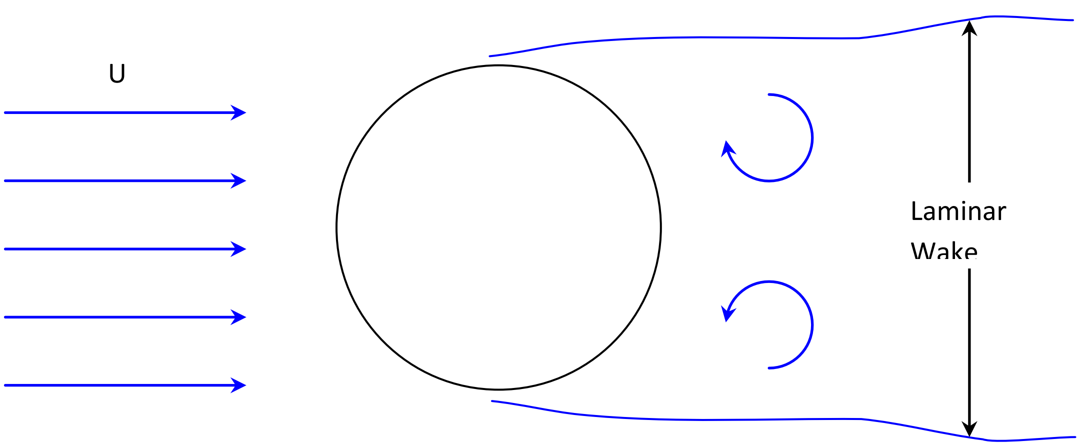
However, fluid pressure field still naturally pushes the particles near the surface to proceed toward B. This is not the case between B and C though, where the natural tendency of the fluid is to flow C to B due to the adverse pressure gradient. Thus the boundary layer slow down that started in the region A to B due to viscous effects bringing C_f toward 0, **gets compounded by the "reverse push" due to the adverse pressure gradient** in the region B to C**. This brings the flow to separation.** **Flow separation point is defined as the point on the surface where C_f = 0, or, τ_w = 0, or** $\left.\frac{\partial u}{\partial y}\right|_{y=0} = 0$.

At flow separation, fluid particles rest on the solid surface but there is **nohold** on them due to shear from the surface. **There is however shearing action from the high-speed flow a little away from the surface, which drags these stagnant particles away into the main flow stream due to viscosity.** This



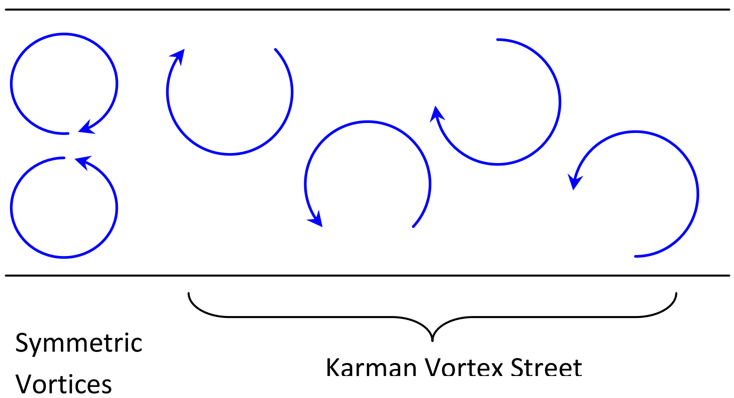
creates a partial void inside the boundary layer, which is promptly filled by particles traveling upstream creating a “reverse flow” near the surface.

The figure shows real flow separation over a circular cylinder with the separation point and reverse flow after separation. Due to symmetry, the exact same processes are repeated on the lower surface ADC. **The reverse flow near the surface is the cause of vortex formation.** Two symmetric vortices appear first in the downstream of the cylinder following flow separation.



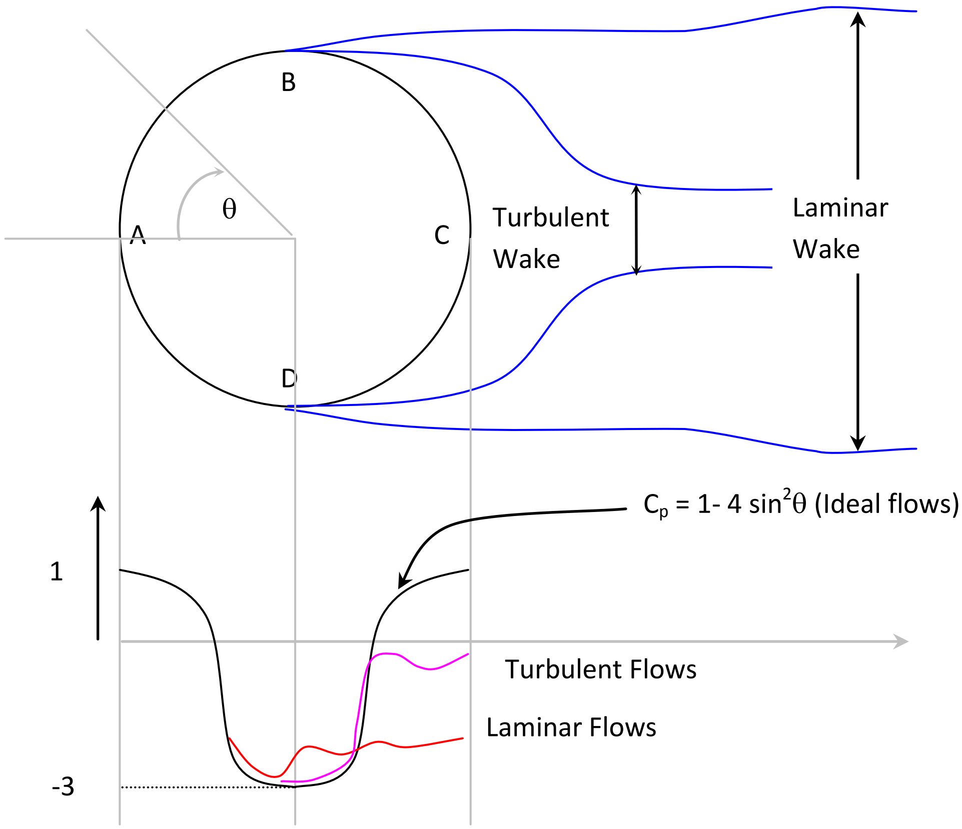
Real Flow Over the Cylinder

These vortices occupy the wake region since they are shed behind the cylinder due to the forward fluid motion. As that process happens the shed vortices grow in size and start interacting with each other creating an **alternating vortex pattern** known as the **Karman Vortex Street**. These create oscillatory flows behind the cylinder.

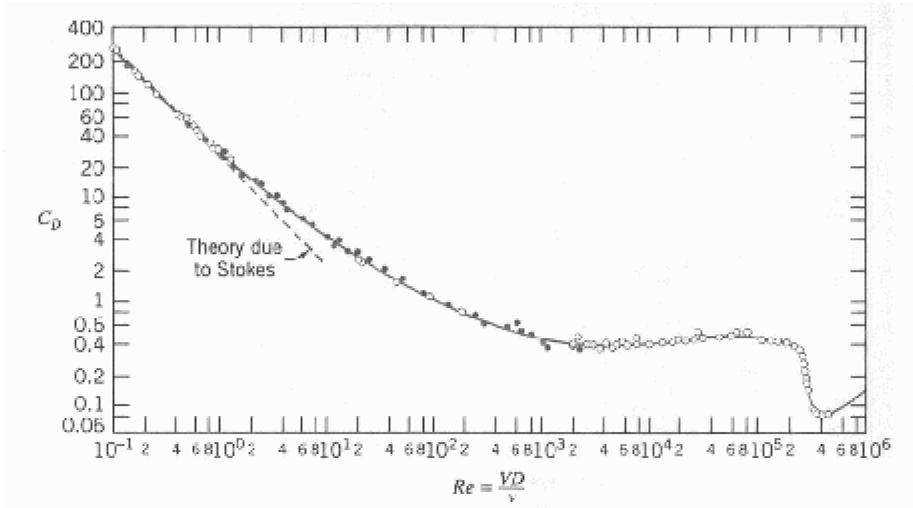


Eventually all the vortices break down due to viscous interactions creating a region of chaos, which is characteristic of a turbulent mixing.

In the initial phase **a laminar separated flow is not necessarily turbulent.** It creates **a large region of low pressure behind the body** called the **wake region.** Due to the separation process, the pressure never recovers its stagnation value in laminar separated flows. If instead of a laminar follow, we had placed the cylinder in a **turbulent flow**, separation will occur with a **much narrower wake** behind the body. This is due to the fact that turbulent flows have **flatter velocity profileswith rapid mixing** and **a lot more momentum** in the boundary layer. **This gives turbulent flows much better chance to resist separation in the region behind the body** (B to C or, D to C). The late separation gives a much smaller wake size with a much better pressure recovery as shown in the figure below:



Thus the drag calculated in the turbulent flows will be much smaller compared to laminar flows (Recall that ideal flow drag is zero due to 100% pressure recovery). This is the reason a turbulent flow separation is preferred over a laminar flow separation (see example of flow momentum calculation). The drag coefficient versus Reynolds number for the flow over a sphere is shown below.



The figure shows that drag coefficient drops as the Reynolds number increases in the low speed range. **In this range, drag on the sphere is directly proportional to the diameter of the sphere ($F_D = 3\pi\mu VD$) as was shown by Stokes.** On the other hand, for high-speed flows, $F_D = \frac{1}{2}\rho U^2 A_b \bullet C_D$. Thus, if C_D is constant, $F_D \propto U^2$. In the low speed range, drag on the sphere is mostly due to friction, whereas in the high-speed range drag is mostly (due to flow separation) from the **pressure drag**. **The sharp drop in the C_D curve around $Re_D = 2x10^5$ is due to the transition from laminar to turbulent flows.** As we saw earlier, transition into turbulence brings smaller wake size and a lower overall drag. This feature is often incorporated into design. For example, golf balls are dimpled to take advantage of this fact. The dimples cause early tripping of the flow into turbulence. This would reduce the drag and will produce longer flights of the ball.

Drag reduction is an active design topic for aerodynamicists and fluid mechanists. A major controlling feature of laminar flow separation is **by removal of stagnant fluid particles near the walls by suction**. Similarly by **blowing into boundary layer, we may be able to energize the stagnant particles and prevent separation**. Control of separation and drag reduction in various applied problems is an active area of research.

Turbulent Boundary Layers

We know that turbulent flow occurs if the flow velocity is large enough (or, viscosity is small enough) to create a Reynolds number greater than the critical Reynolds number over an object. For spheres or circular cylinders this critical Reynolds number is between 2 to $4x10^5$. For flat plate flows this is around 500,000. We also discussed the implications of turbulent flows in drag reduction. What characterizes such flow is a flatter, fuller velocity profile. It is important to recognize that turbulent flows have two components: (i) a mean, $\bar{u}$, and (ii) a random one, u′. $\therefore u = \bar{u} + u'$. Similarly, $v = \bar{v} + v'$ and $w = \bar{w} + w'$. The random u′ cannot be determined without statistical means. Therefore for turbulent fluid flows, we usually work with a timeaveraged mean flow $\bar{u}$. Remember that when we speak of turbulent velocity profiles it is this $\bar{u}$ that we are considering. To avoid confusion with this rotation (we earlier indicated $\bar{v}$ as <u>areaaveraged</u> velocity, not time average velocity), we shall write turbulent flow velocities without the bars.

You understand that whenever we speak about turbulent flows here, we are representing the mean flow. Turbulent flows in boundary layers over flat plates may be represented by the power law velocity profile:

$$\frac{u}{U}(y) = \left(\frac{y}{\delta}\right)^{1/n} = \eta^{1/n} \quad \left[\text{where, } \eta = \frac{y}{\delta}\right]$$

This profile covers a fairly broad range of turbulent Reynolds numbers for 6 < n <10. The most popular one is n = 7. Although this velocity profile is an excellent representation of the real turbulent flow, this may not be used to calculate skin friction coefficient in the approximate solution method seen earlier (since $\left.\frac{\partial u}{\partial y}\right|_{y=0}$ will be negligible for this profile). For the purpose of calculating shear stress we use an experimental result: $\tau_w = 0.0233\,\rho U^2\left(\frac{\nu}{U\delta}\right)^{1/4}$ for the 1/7 power law profile. To obtain the skin friction coefficient, we must first evaluate δ(x) from the solution of Karman-Pohlhausen:

$$\tau_w = \rho U^2 \frac{\partial \Theta}{\partial x} \qquad \text{(A)}$$

$$\because\ \Theta = \int_0^\delta \frac{u}{U}\left(1-\frac{u}{U}\right)dy \Rightarrow \frac{\Theta}{\delta} = \int_0^1 \frac{u}{U}\left(1-\frac{u}{U}\right)dy$$

$$\text{Using } \frac{u}{U} = \eta^{1/7},\ \frac{\Theta}{\delta} = \int_0^1 \eta^{1/7}\left(1-\eta^{1/7}\right)d\eta$$

$$\int_0^1\left(\eta^{\frac{1}{7}} - \eta^{\frac{2}{7}}\right)d\eta = \frac{7}{8} - \frac{7}{9} = \frac{7}{72}$$

$$\therefore (A) \Rightarrow \tau_w = 0.0233\rho U^2\left(\frac{\nu}{U\delta}\right)^{1/4} = \rho U^2\left(\frac{7}{72}\right)\frac{d\delta}{dx}$$

$$\text{or, } \int_{\delta=0}^{\delta(x)} \delta^{1/4} d\delta = \int_{x=0}^{x} .24\left(\frac{\nu}{U}\right)^{1/4} dx$$

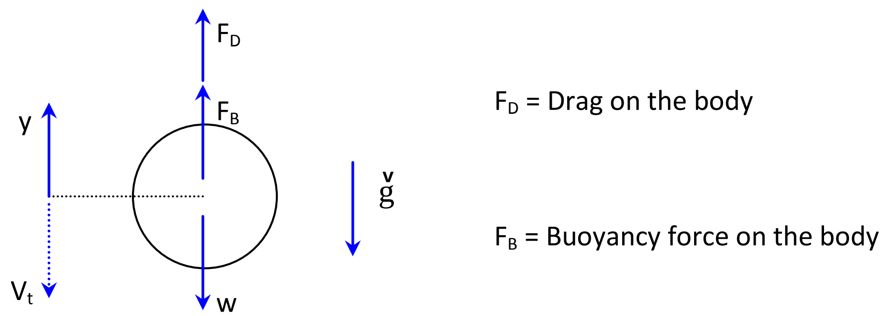
$$\Rightarrow \boxed{\frac{\delta}{x} = \frac{.382}{Re_x^{1/5}}} \qquad \text{(Skipping the integral evaluation)}$$

Terms for the 1/7 power law velocity profile gives:

$$\boxed{C_f = \frac{.0594}{Re_x^{1/5}}}$$

<u>**Stokes Flows**</u>

We have so far discussed very high-speed flows in which the boundary layers are very thin regions near the body. However for very low speed flows boundary layers don't exist. **Viscous effects are felt everywhere (recall the <u>heat transfer analogy</u>).** External flow applications at very slow speeds (or, highly viscous flows) may be solved by neglecting the inertia force term in the **<u>Newton's second law</u>.** For example, if you drop a steel ball into glycerin, how can you calculate drag on it? In this context, let us introduce the concept of terminal velocity. When any object starts its motion in any fluid medium, there may be a period of acceleration of motion. However, if we are interested in steady flows, if one exists in such configurations**, there must be a time when the fluid forces around the body are balanced providing it a constant velocity. We call this velocityterminal velocity of the body**. For the ball dropped in glycerin, the free body diagram shows



Note that the V_t (Terminal velocity) is not a force, and shown on the sketch (just for reference) using dashed lines. Since the body is traveling at constant speed, the inertia force term is zero. Thus, all external forces are balanced and in the Newton's second law in the y-direction:

$$\sum F_{B_y} + \sum F_{S_y} = ma_y = 0 \quad \ldots\ldots\ldots\ldots\ldots\textbf{(A)}$$

$$\Rightarrow W - F_B - F_D = 0$$

where,

$$W = mg$$

$$F_B = \rho g \forall$$

$$F_D = 3\pi\mu V_t D$$

$$\forall = \frac{\pi D^3}{6}$$

D = Diameter, ρ = Density of glycerin
g = Acceleration due to gravity

We may only use the above equation to calculate the terminal velocity V_t.

In the above <u>drag representation</u> of Stokes flow, $F_D \propto V_t$. This behavior is in contrast with high-speed flows, where drag is usually proportional to the square of velocity.

Terminal velocity concept is similar to fully developed flows in internal flow configuration. Notice that before the terminal velocity is developed (in the internal flow case, in the entrance length region), the inertia force term in the above equation (A) is not negligible. In that case, the only way to solve the equation will be by integration or, using differential equations approach

For engineering design purposes, handbooks list a large variety of objects in different orientations and their drag coefficients. Rather than solving each problem from first principles, you may be able to utilize these tables and charts. Just make sure that you note the range of applicability of these. They need to be verified during problem solving.