PHY2009S

"Fields and fluids"

Part B: Introduction to fluid mechanics

Andy Buffler

Department of Physics

University of Cape Town

Bibliography

Fundamentals of fluid mechanics, 4th Edition, Munson, Young and Okiishi, (Wiley, 2002)

A physical introduction to fluid mechanics, Smits, (Wiley, 2000)

Introduction to fluid mechanics, 5th Edition, Fox and McDonald, (Wiley, 1998)

Fluid mechanics, 4th Edition, Douglas, Gasiorek and Swaffield, (Prentice Hall, 2001)

The Feynman lectures on physics, Volume II, Feynman, Leighton and Sands, (Addison-Wesley, 1964)

Fluid dynamics for physicists, Faber, (Cambridge, 1997)

Physical fluid dynamics, Tritton, (Van Nostrand, 1977)

Video: "The mechanical universe and beyond"
No. 36, *Vector fields and hydrodynamics*

What do you remember from Physics I ?

(Properties of matter and thermodynamics)

- Force
- Pressure
- Density
  - Specific volume
  - Specific weight
- Stress and strain
- Young's modulus
- Bulk modulus
- Viscosity
- Surface tension
- Vapour pressure
- Thermodynamics: First Law, Second Law, …
- Heat capacity
- Dimensional analysis.

Hydrostatistics

(The theory of liquids at rest)

A fluid cannot maintain a sheer stress for any length of time. The measure of ease with which the fluid yields is given by its **viscosity**.

It is complicated to include viscosity in our model
$\rightarrow$ first ignore viscous effects (frictionless or inviscid flow)

When liquids are at rest there are no shear forces. In a static fluid the force per unit area across any surface is normal to the surface and is the same for all orientations of the surface.

Hydrostatistics ... 2

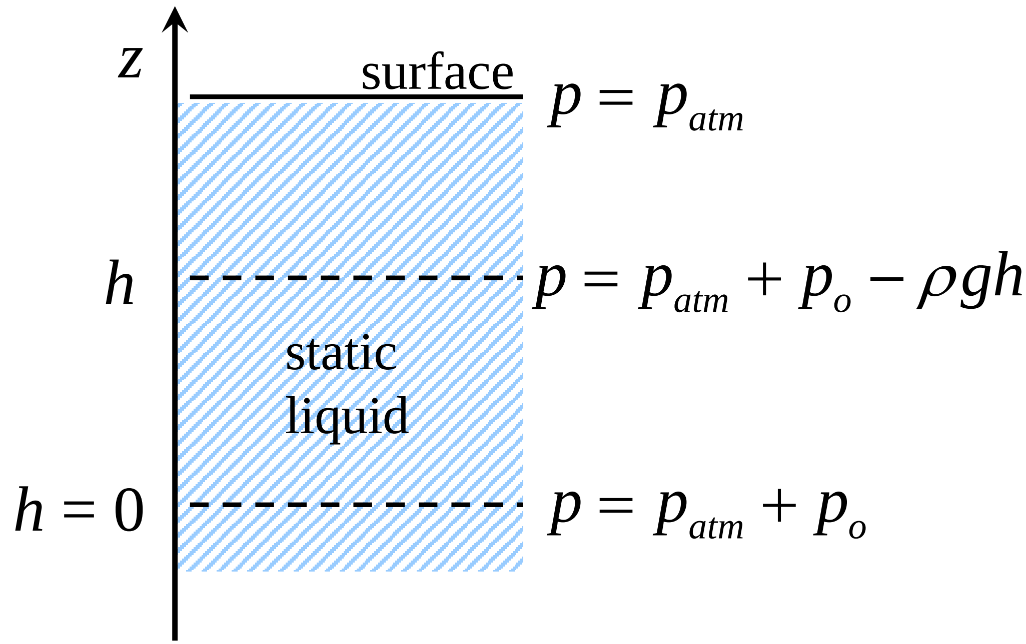
The normal force per unit area is the pressure, p.

In a static fluid the pressure must be the same in all directions at a particular point in the liquid.

The pressure, of course, may vary from point to point in the liquid.

For example, in a static liquid on the earth's surface:

ρ : density (constant)



p_{atm} : atmospheric pressure

p_o : "gauge" pressure (pressure measured above p_{atm})

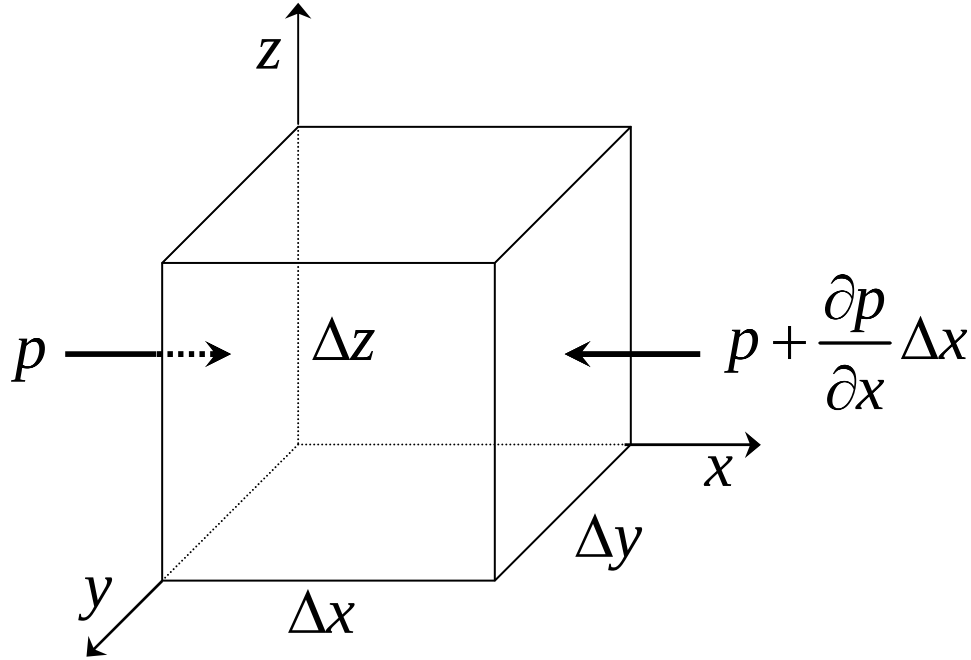
"Absolute" pressure = atm pressure + gauge pressure = $p_{atm} + p_o$

Video clip (2.3) A dam wall: For an incompressible fluid at rest the pressure increases linearly with the fluid depth. As a consequence, large forces can be developed on plane and curved surfaces in contact with the fluid. The Hoover Dam, on the Colorado river, is the highest concrete arch-gravity type of dam in the United States. The water behind Hoover dam is approximately 220 m deep and at this depth the pressure is 21 atm. To with-stand the large pressure forces on the face of the dam, its thickness varies from 15 m at the top to 200 m at the base.

Video Clip (3.5) Flow from a tank: According to the Bernoulli equation, the velocity of a fluid flowing through a hole in the side of an open tank or reservoir is proportional to the square root of the depth of fluid above the hole.
The velocity of a jet of water from an open cooldrink bottle containing four holes is clearly related to the depth of water above the hole. The greater the depth, the higher the velocity.

Hydrostatistics … 3

Consider now a small cube of fluid with pressure varying in the *x*-direction only.



Pressure on the face at position *x* gives the force: $p\Delta y\Delta z\hat{\mathbf{i}}$

Pressure on the face at $x+\Delta x$ gives the force: $-\left[p+\dfrac{\partial p}{\partial x}\Delta x\right]\Delta y\Delta z\hat{\mathbf{i}}$

Resultant pressure force on the cube in the *x*-direction: $-\dfrac{\partial p}{\partial x}\Delta x\Delta y\Delta z\hat{\mathbf{i}}$

… and similarly for the other faces ...

Resultant pressure force on the cube in the *y*-direction: $-\dfrac{\partial p}{\partial y}\Delta x\Delta y\Delta z\hat{\mathbf{j}}$

Resultant pressure force on the cube in the *z*-direction: $-\dfrac{\partial p}{\partial z}\Delta x\Delta y\Delta z\hat{\mathbf{k}}$

Hydrostatistics ... 4

Total resultant pressure force on the cube = $-\left(\frac{\partial p}{\partial x}\hat{\mathbf{i}}+\frac{\partial p}{\partial y}\hat{\mathbf{j}}+\frac{\partial p}{\partial z}\hat{\mathbf{k}}\right)\Delta x\Delta y\Delta z$

Pressure force per unit volume = $-\vec{\nabla}p$

There could be other forces acting on the cube of liquid, such as gravity. Assume that the gravitational force can be described in terms of a potential energy.

Let ϕ be the potential energy per unit mass

For gravity, for example, $\phi = gz$

Then the force per unit mass = $-\vec{\nabla}\phi$

and the force per unit volume = $-\rho\vec{\nabla}\phi$

Then for equilibrium: $-\vec{\nabla}p-\rho\vec{\nabla}\phi = 0$

Equation of hydrostatics

Hydrostatistics … 5

In general, $-\vec{\nabla} p - \rho \vec{\nabla} \phi \; = \; 0$ has no solution.

… since if the density ρ varies in space in some arbitrary way, then there is no way for the forces to be in balance, and the fluid cannot be in static equilibrium. **Convection currents** will start up.

Only when ρ is constant is the potential term a pure gradient and the equation has a solution:

$$p + \rho\phi = \text{ constant}$$

(a solution is also possible if ρ is a function of only p)

Liquids at rest are boring!

... it is much more interesting when

the liquid is moving …

Fluid kinematics

Need both vector and scalar quantities

Steady flow: when the flow pattern and speed at each position remains unchanged over a given time period.

Steady flow doesn't exist in nature, but is a good approximation in many situations.

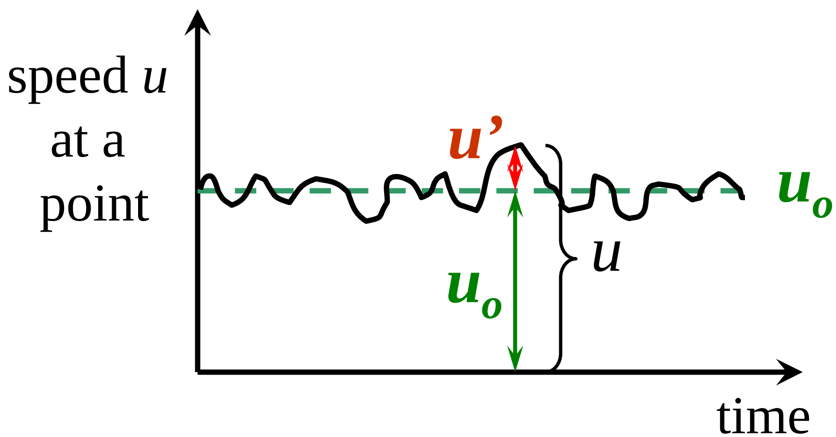
… so we use "quasi-steady flow"

$$u = u_o + u'$$

time-dependent part (u_o)

fluctuations (u')

$$u_o = \frac{1}{\tau}\int_{t-\frac{\tau}{2}}^{t+\frac{\tau}{2}} u \, dt$$



τ : averaging time chosen to be appropriate for the problem.

Steady and unsteady flow

In reality, almost all flows are **unsteady** in some sense.
Unsteady flows are usually more difficult to analyze and to investigate experimentally than are steady flows. Among the various types of unsteady flows are **non-periodic** flow, **periodic** flow and truly **random** flow.

An example of a **non-periodic** unsteady flow is that produced by turning off a tap to stop the flow of water. Usually this unsteady flow process is quite mundane and the forces developed as a result of the unsteady effects need not be considered. However, if the water is turned off suddenly, as with an electrically operated valve in a dishwasher, the unsteady effects can become important [as in the "water hammer" effects made apparent by the loud banging of the pipes under such conditions].

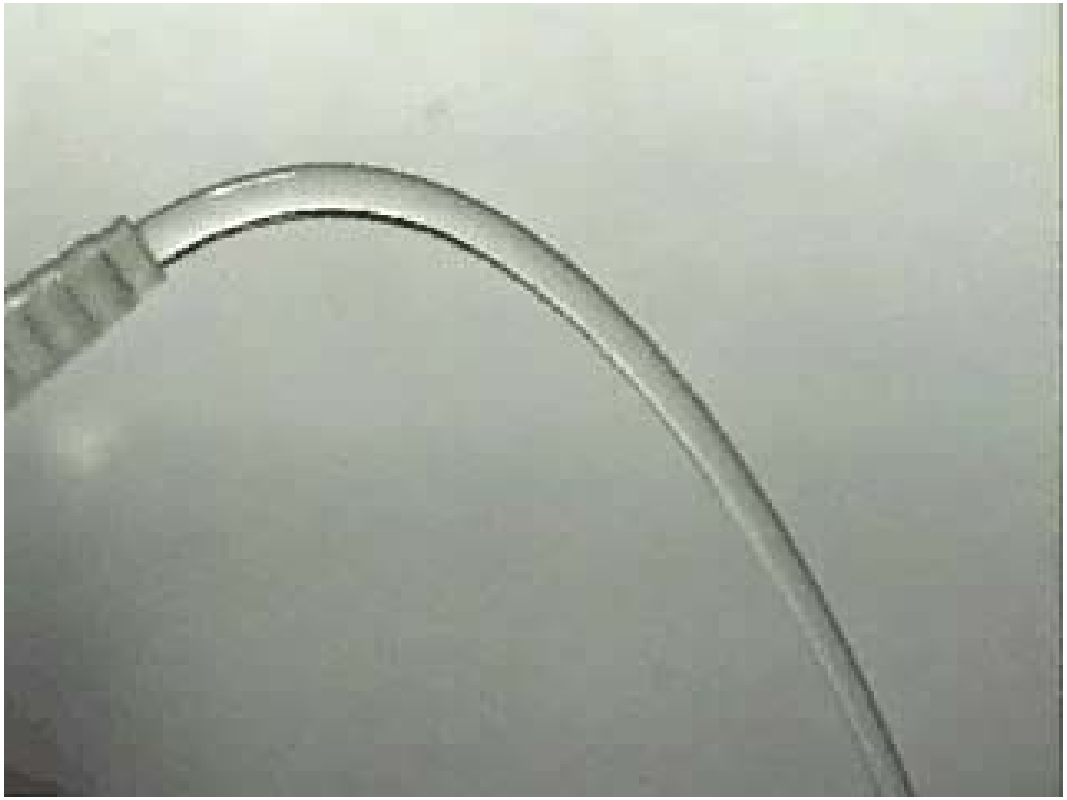
Steady and unsteady flow ... 2

In other flows the unsteady effects may be **periodic**, occurring time after time in basically the same manner. The periodic injection of the air-gasoline mixture into the cylinder of an automobile engine is such an example. The unsteady effects are quite regular and repeatable in a regular sequence. They are very important in the operation of the engine.

In many situations the unsteady character of a flow is quite **random**. There is no repeatable sequence or regular variation to the unsteadiness. This behaviour occurs in **turbulent flow** and is absent from **laminar flow**.

The “smooth” flow of highly viscous syrup represents a “deterministic” laminar flow.

It is quite different from the turbulent flow observed in the “irregular” splashing of water from a faucet onto the sink below it. The “irregular” gustiness of the wind represents another random turbulent flow.



Video clip (4.3) Flow types: Among the many ways that flows can be categorized are: a) steady or unsteady, b) laminar or turbulent. Which type of flow occurs for a given situation depends on numerous parameters that affect the flow. The low speed flow of water from a small nozzle is steady and laminar. Unless the flow is disturbed (by poking it with a pencil, for example), it is not obvious that the fluid is moving. On the other hand, the flow within a clothes washer is highly unsteady and turbulent. Such motion is needed to produce the desired cleaning action.

Fluid kinematics ... 2

A typical portion of fluid contains so many molecules that it becomes totally unrealistic, except in special cases, for us to attempt to account for the motion of individual molecules. Rather, we employ the **continuum hypothesis** and consider fluids to be made up of fluid particles that interact with each other and with their surroundings. Each particle contains numerous molecules. Thus, we can describe the flow of a fluid in terms of the motion of **fluid particles** rather than individual molecules.

The infinitesimal particles of a fluid are tightly packed together. Thus, at a given instant in time, a description of any fluid property such as density, pressure, velocity, and acceleration, may be given as a function of the fluid's location. This representation of fluid parameters as functions of the spatial coordinates is termed a **field representation** of the flow.

Fluid kinematics ... 3

In order to describe the motion of a fluid-need to describe its properties at every point: velocity, acceleration, pressure, density, temperature, electric current, magnetic field…

We need to specify each property as function of (x,y,z) and time t.

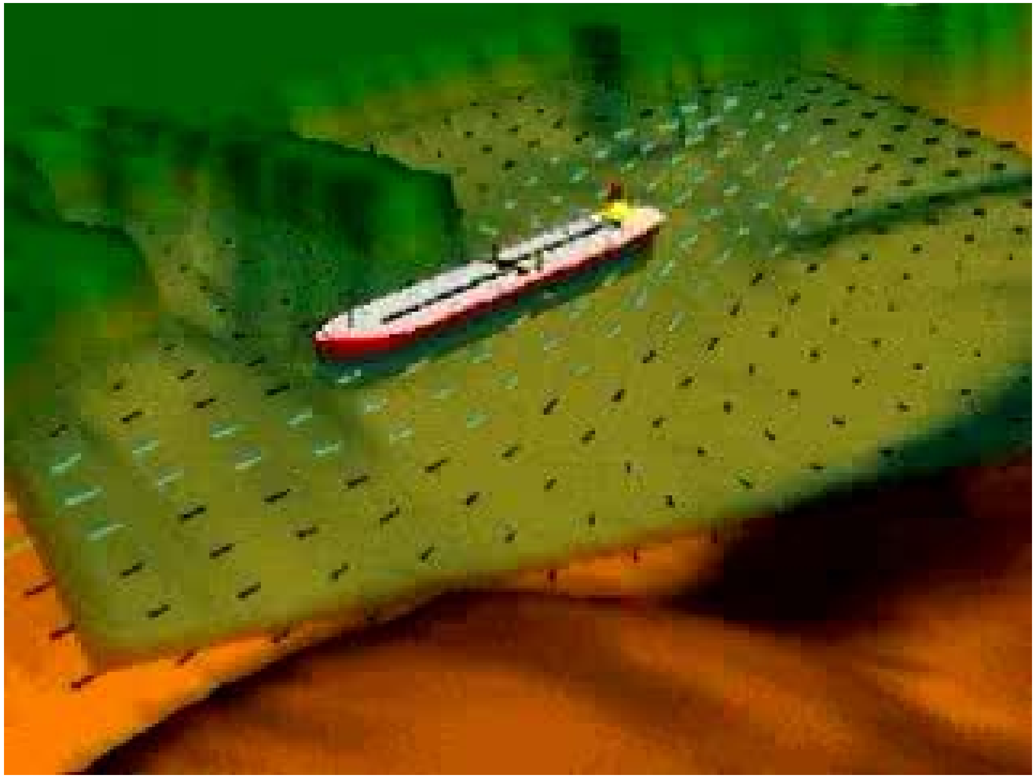
One of the most important fluid variables is the **velocity field:**

In Cartesian coordinates:

$$\vec{\mathbf{v}}\left(x,y,z,t\right)=v_x(x,y,z,t)\hat{\mathbf{i}}+v_y(x,y,z,t)\hat{\mathbf{j}}+v_z(x,y,z,t)\hat{\mathbf{k}}$$

or in cylindrical coordinates:

$$\vec{\mathbf{v}}\left(r,\theta,z,t\right)=v_r(r,\theta,z,t)\hat{\mathbf{r}}+v_y(r,\theta,z,t)\hat{\boldsymbol{\theta}}+v_z(r,\theta,z,t)\hat{\mathbf{z}}$$



<u>Video clip (4.1) A Velocity Field</u>: A field representation is often used to describe flows. In doing so, the flow parameters are specified as functions of the spatial coordinates and time. The velocity field describes a flow by giving the point-by-point fluid velocity throughout the flow field. The calculated velocity field in a shipping channel is shown as the tide comes in and goes out. The fluid speed is given by the length and color of the arrows. The instantaneous flow direction is indicated by the direction that the velocity arrows point.

Fluid kinematics ... 4

In almost any flow situation, the velocity field actually contains all three velocity components.
In many situations the **three-dimensional** flow characteristics are important in terms of the physical effects they produce. For these situations it is necessary to analyze the flow in its complete three-dimensional character.
The flow of air past an airplane wing provides an example of a complex three-dimensional flow.

In many situations one of the velocity components may be small relative to the two other components.
In situations of this kind it may be reasonable to neglect the smaller component and assume **two-dimensional flow**, which is simpler to deal with both pictorially and analytically.

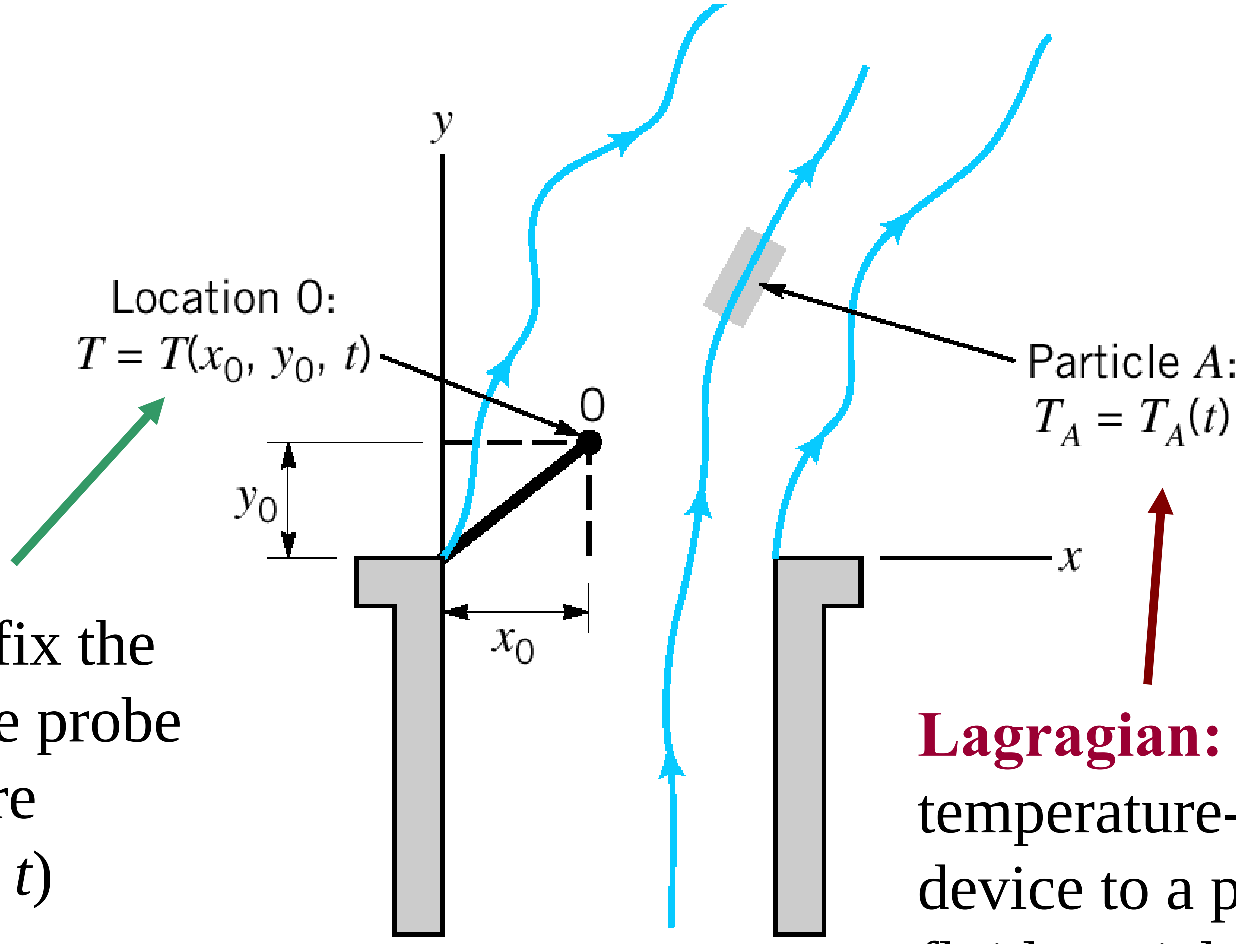
Eulerian and Lagrangian Flow Descriptions

There are two general approaches in analyzing fluid mechanics problems:

Eulerian method: uses the field concept introduced above. In this case, the fluid motion is given by completely prescribing the necessary properties (pressure, density, velocity, etc.) **as functions of space and time**. From this method we obtain information about the flow in terms of what happens at **fixed points in space** as the fluid flows past those points.

Lagrangian method: involves following **individual fluid particles** as they move about and determining how the fluid properties associated with these particles change as a function of time. The fluid particles are “tagged ”or identified, and their properties determined as they **move**.

Studying the temperature of smoke from a chimney:



Eulerian: fix the temperature probe and measure $T(x_o, y_o, z_o, t)$

Lagragian: attach the temperature-measuring device to a particular fluid particle (A) and record that particle's temperature as it moves about.

In fluid mechanics it is usually **easier** to use the Eulerian method to describe a flow in either experimental or analytical investigations. There are, however, certain instances in which the **Lagrangian method** is more convenient.

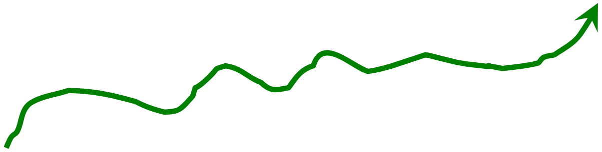
For example, some numerical fluid mechanics calculations are based on determining the motion of individual fluid particles, based on the appropriate interactions among the particles, thereby describing the motion in Lagrangian terms. Similarly, in some experiments individual fluid particles are “tagged” and are followed throughout their motion

For example, oceanographic measurements obtained from devices that flow with the ocean currents provide this information. Similarly, by using X-ray opaque dyes it is possible to trace blood flow in arteries.

A Lagrangian description may also be useful in describing fluid machinery, such as pumps and turbines, in which fluid particles gain or lose energy as they move along their flow paths.

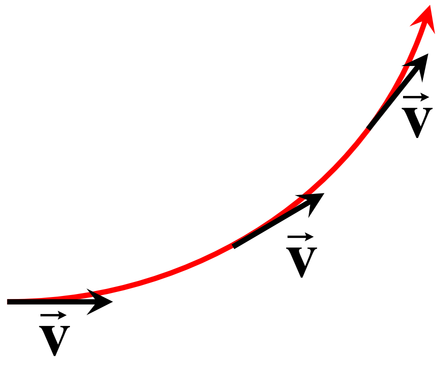
Some useful terms and constructions for fluid dynamics:

Pathline: the path traced out by a single particle over a period of time.



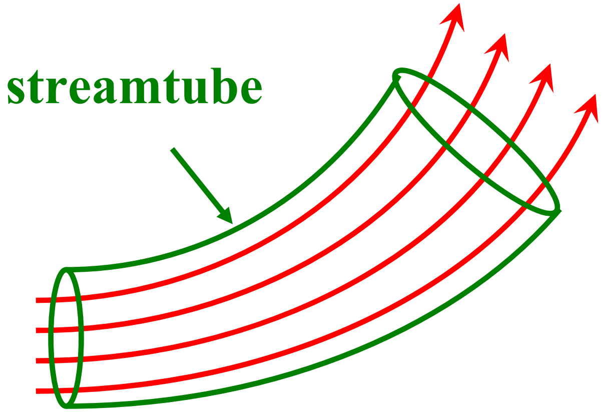
The pathline is a Lagrangian concept that can be produced in the laboratory by marking a fluid particle (dying a small fluid element) and taking a time exposure photograph of its motion.

Streamline: line joining the tangent of instantaneous velocity vectors of the whole fluid as a function of position in the fluid-form an instantaneous flow pattern.

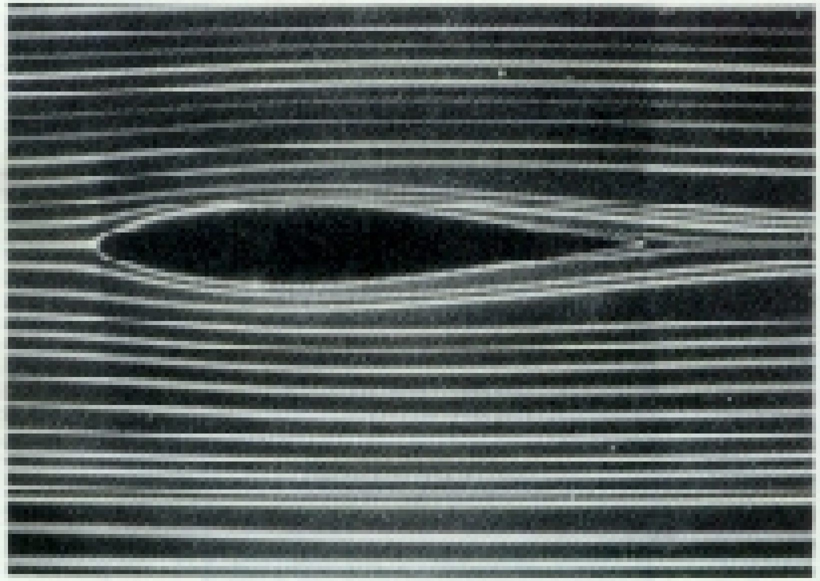


If the flow is steady, the streamlines are fixed lines in space. For unsteady flows the streamlines may change shape with time. Streamlines are obtained analytically by integrating the equations defining lines tangent to the velocity field. For 2D flows the slope of the streamline dy/dx must be equal to the tangent of the angle that the velocity vector makes with the *x*-axis or $\frac{dy}{dx} = \frac{v_y}{v_x}$.

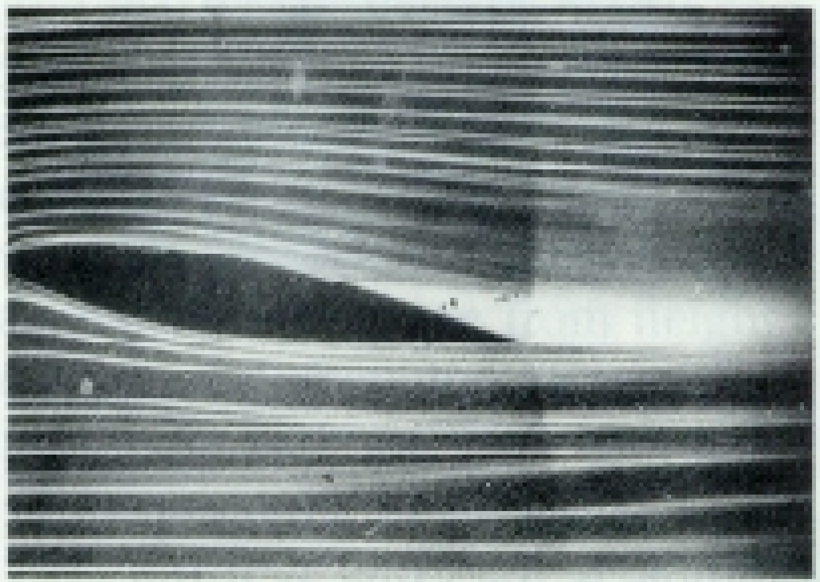
Streamtube: a selection of streamlines considered together. There can be no flow across the surface of the streamtube.



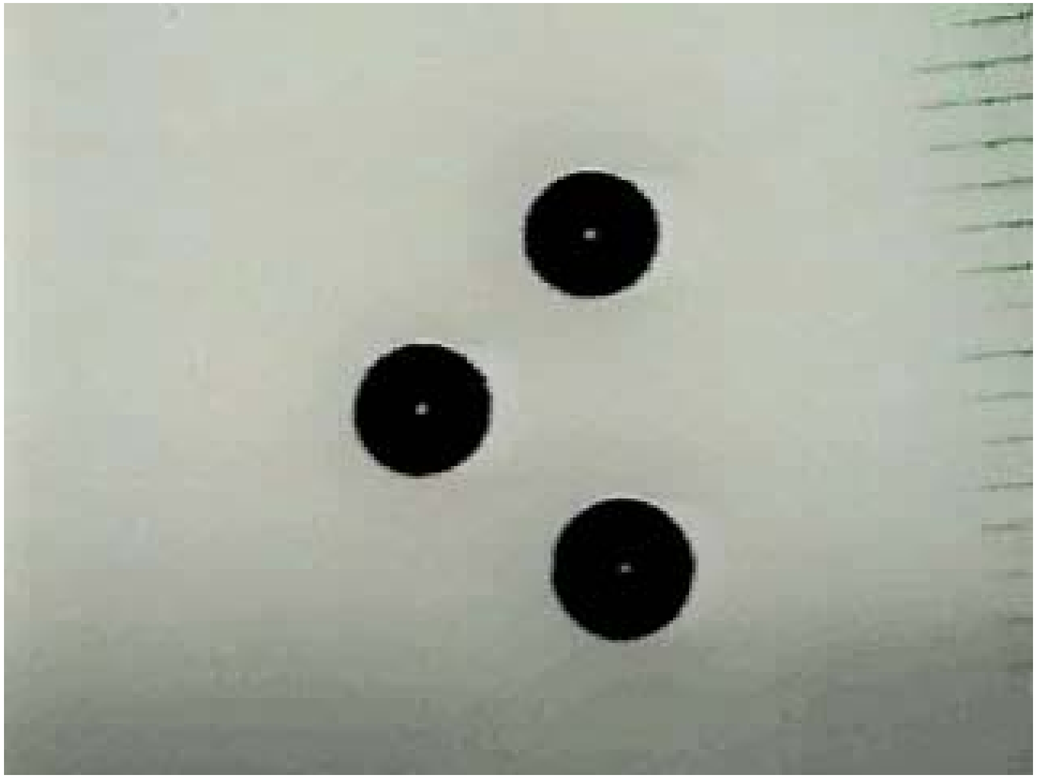
Stream filament: an infinitesimally small streamtube.



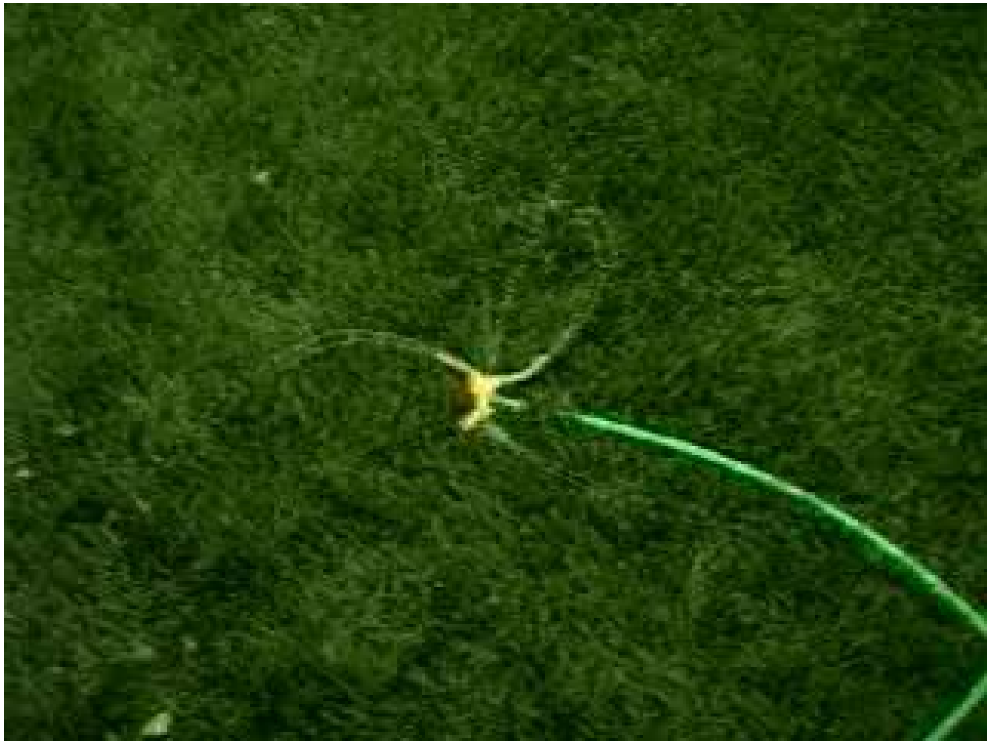
(*a*)



(*b*)

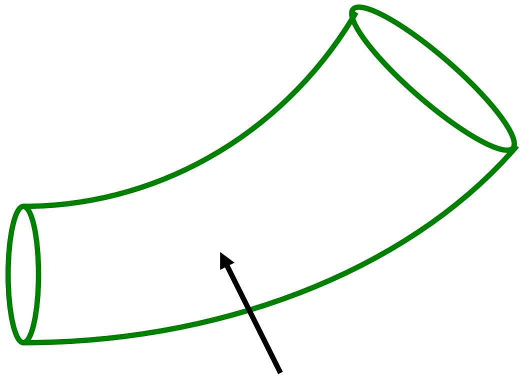


<u>Video clip (4.5) Streamlines:</u> A streamline is a line that is everywhere tangent to the velocity field. For steady flow, the streamlines are fixed lines in a flow field which show the paths the fluid particles follow. Streamlines created by injecting dye into water flowing steadily around a series of cylinders reveal the complex flow pattern around the cylinders. Also, as shown for flow around a series of airfoil shapes, the stagnation streamline and corresponding stagnation point can be easily observed (see top airfoil). Stagnation points on the other airfoils could be located by adjusting the upstream position of the incoming streamlines.



<u>Video clip (4.6) Pathlines:</u> A pathline is the line traced out by a given particle as it flows from one point to another. The lawn sprinkler rotates because the nozzle at the end of each arm points “backwards”. This rotation give the water a “forward” component. As a result, water particles leave the end of each spray arm in a nearly radial direction. Once the water exits the spray arm there is no external force to change its velocity (neglecting aerodynamic drag and gravity). Thus, the pathlines are essentially straight radial lines, whereas the shape of the water streams (which are not pathlines) is a complex spiral about the axis of rotation.

Stream surface: the surface of a stream tube, often used to illustrate an interface between two different regimes of flow.



Streakline: a line connecting all the particles that have emanated from a single point in a flow field.

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

For steady flow, streamlines, streaklines, and pathlines are the same.

The equation of continuity

Consider a situation where there is no magnetic field and no conductivity and the temperature is governed by density and pressure.
Also assume that the density is the same at all points in the fluid (i.e. fluid is incompressible)

The equation of state is simply: $\rho = \text{constant}$

If matter flows away from a point, there must be a decrease in the amount left behind. The mass of fluid which flows in a unit time across a unit area is the component of $\rho\vec{\mathbf{v}}$ normal to the surface, where $\vec{\mathbf{v}}$ is the fluid velocity.

Hydrodynamic equation of continuity: $\vec{\nabla}\bullet(\rho\vec{\mathbf{v}}) = -\frac{\partial\rho}{\partial t}$

(an expression of the conservation of mass of the fluid.)

The equation of continuity …2

Hydrodynamic equation of continuity: $\vec{\nabla}\bullet\left(\rho\vec{\mathbf{v}}\right) = -\dfrac{\partial\rho}{\partial t}$

Since ρ = constant, then $\dfrac{\partial\rho}{\partial t} = 0$

and our **equation of continuity** becomes: $\vec{\nabla}\bullet\vec{\mathbf{v}} = 0$

(the fluid velocity $\vec{\mathbf{v}}$ has zero divergence)

The streamlines must be **solenoidal**: they either form closed curves or terminate on a boundary or at infinity. This is analogous to the magnetic field lines around a current-carrying wire.

Example:

For a steady, two-dimensional flow field in the *xy*-plane, the *x*-component of velocity is given by $v_x = Ax$.
Determine a possible *y*-component for incompressible flow.

Example:

The velocity components for a certain steady, three-dimensional flow field are

$$v_x = x^2 + y^2 + z^2.$$
$$v_y = xy + yz + z$$
$$v_z = ?$$

Determine v_z.

The stream function

Steady, incompressible, plane, two-dimensional flow represents one of the simplest types of flow of practical importance.
For this flow, the continuity equation $\vec{\nabla} \bullet \vec{\mathbf{v}} = 0$ reduces to

$$\frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} = 0$$

We still have two variables v_x and v_y to deal with, but they must be related in a special way.
The continuity equation suggests that we can define a function $\psi(x, y)$ called the **stream function** which relates the velocities as

$$v_x = \frac{\partial \psi}{\partial y} \quad \text{and} \quad v_y = -\frac{\partial \psi}{\partial x}$$

The stream function ... 2

Another particular advantage of using the stream function is related to the fact that lines along which ψ is constant are **streamlines**. Recall that streamlines are lines in the flow field that are everywhere tangent to the velocities. It follows from the definition of the streamline that the slope at any point along a streamline is given by

$$\frac{dy}{dx} = \frac{v_y}{v_x}$$

The change in the value of ψ as we move from one point (x,y) to a nearby point (x+dx, y+dy) is given by the relationship:

$$d\psi = \frac{\partial \psi}{\partial x} dx + \frac{\partial \psi}{\partial y} dy = -v_y dx + v_x dy$$

The stream function ... 3

Along a line of constant ψ we have $d\psi = 0$ so that

$$-v_y dx + v_x dy = 0$$

and, therefore, along a line of constant ψ:

$$\frac{dy}{dx} = \frac{v_y}{v_x}$$

which is the defining equation for a streamline.

Thus, if we know the function we can plot lines of constant ψ to provide the family of streamlines that are helpful in visualizing the particular pattern of flow being studied.

Example:

A steady, incompressible, two-dimensional flow field is described by:

$$\vec{\mathbf{v}}(x, y) = 2y\hat{\mathbf{i}} + 4x\hat{\mathbf{j}} \quad \text{m s}^{-1}$$

Determine the stream function and show on a sketch several streamlines.

The acceleration field

Newton II: the mass of an element of volume multiplied by its acceleration equals the force on that volume element

For an element of unit volume: $\rho \times \vec{\mathbf{a}} = \vec{\mathbf{F}}$

force per unit volume

The pressure force per unit volume = $-\vec{\nabla} p$

External forces (e.g. gravity) gives a force density: $-\rho \vec{\nabla} \phi$

(where ϕ is the potential per unit mass)

There may also be "internal" forces per unit volume due to that in a flowing fluid we can also be a shearing stress, called the viscous force: $\vec{\mathbf{f}}_{\mathbf{visc}}$

The acceleration field ... 2

Putting this all together gives:

Equation of motion: $\rho\vec{\mathbf{a}} = -\vec{\nabla}p - \rho\vec{\nabla}\phi + \vec{\mathbf{f}}_{\mathrm{visc}}$

We are studying ideal, "dry" fluids (which is totally unnatural)

Ignore

Setting $\vec{\mathbf{f}}_{\mathrm{visc}} = 0$ is leaving out an **essential** property of a fluid.

Our equation of motion becomes $\rho\vec{\mathbf{a}} = -\vec{\nabla}p - \rho\vec{\nabla}\phi$

This is called the Euler equation, or the momentum equation for inviscid flow.
If $\vec{\mathbf{f}}_{\mathrm{visc}}$ is included, then it becomes the Navier-Stokes equation (see later)

The Euler equation

Equation of motion: $\rho\vec{\mathbf{a}} = -\vec{\nabla}p - \rho\vec{\nabla}\phi$

Is $\vec{\mathbf{a}} = \dfrac{\partial\vec{\mathbf{v}}}{\partial t}$? ... No !

$\dfrac{\partial\vec{\mathbf{v}}}{\partial t}$ is the rate of change of $\vec{\mathbf{v}}(x,y,z,t)$ at a fixed point in space.

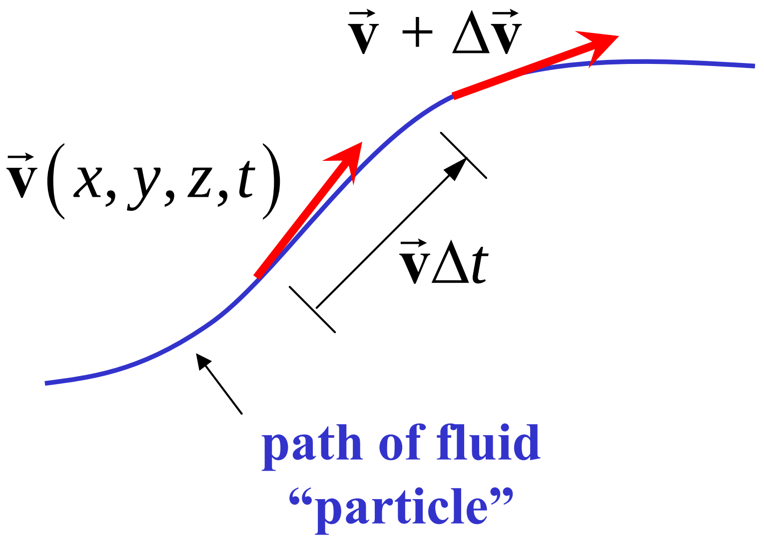
However, we need the rate of change of $\vec{\mathbf{v}}$ for a particular "piece" of fluid.

Velocity of particle at position (x,y,z) at time t = $\vec{\mathbf{v}}(x,y,z,t)$

Velocity of the same particle at time $t + \Delta t$

$= \vec{\mathbf{v}}(x+\Delta x\ ,y+\Delta y\ ,\ z+\Delta z\ ,\ t+\Delta t)$

where $\Delta x = v_x\Delta t\ ,\ \Delta y = v_y\Delta t\ ,\ \Delta z = v_z\Delta t$



The Euler equation ... 2

From the definition of partial derivatives:

$$\vec{\mathbf{v}}\left(x+v_x\Delta t\ ,y+v_y\Delta t\ ,\ z+v_z\Delta t\ ,\ t+\Delta t\right)$$

$$= \ \vec{\mathbf{v}}\left(x,y,z,t\right) + \frac{\partial\vec{\mathbf{v}}}{\partial x}v_x\Delta t + \frac{\partial\vec{\mathbf{v}}}{\partial y}v_y\Delta t + \frac{\partial\vec{\mathbf{v}}}{\partial z}v_z\Delta t + \frac{\partial\vec{\mathbf{v}}}{\partial t}\Delta t$$

The acceleration $\lim\limits_{\Delta t\to 0}\frac{\Delta\vec{\mathbf{v}}}{\Delta t} = \frac{\partial\vec{\mathbf{v}}}{\partial x}v_x + \frac{\partial\vec{\mathbf{v}}}{\partial y}v_y + \frac{\partial\vec{\mathbf{v}}}{\partial z}v_z + \frac{\partial\vec{\mathbf{v}}}{\partial t}$

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

where $\frac{D}{Dt}$ is the "total derivative".

$$\therefore \quad \boxed{\rho\frac{D\vec{\mathbf{v}}}{Dt} = -\vec{\nabla}p - \rho\vec{\nabla}\phi}$$

Euler equation for an inviscid fluid

The total derivative

$$\frac{D}{Dt} \quad = \quad \underbrace{\frac{\partial}{\partial x}v_x + \frac{\partial}{\partial y}v_y + \frac{\partial}{\partial z}v_z}_{} \qquad + \qquad \frac{\partial}{\partial t}$$

"**convective**" part
... describes the variation that occurs in space

"**local**" part
... describes the variation in the field with time at a given point

$\frac{D}{Dt}$ is also known as the "**substantive**", "particle" or "**material**" derivative

(There can be an acceleration even if $\frac{\partial \vec{\mathbf{v}}}{\partial t} = 0$.)

Example:

Given the Eulerian, Cartesian velocity field

$$\vec{\mathbf{v}} = 2t\hat{\mathbf{i}} + zx\hat{\mathbf{j}} + t^2 y\hat{\mathbf{k}} \quad \text{m s}^{-1}$$

find the acceleration following a fluid particle.

Bernoulli's theorem

For steady flow, the velocity of the fluid at every point is constant,

i.e $\dfrac{\partial \vec{\mathbf{v}}}{\partial t} = 0$ and our equation of motion reduces to:

$$\rho\left(\vec{\mathbf{v}}\bullet\vec{\nabla}\right)\vec{\mathbf{v}} = -\vec{\nabla}p - \rho\vec{\nabla}\phi$$

Use the identity: $\left(\vec{\mathbf{A}}\bullet\vec{\nabla}\right)\vec{\mathbf{A}} = \left(\vec{\nabla}\times\vec{\mathbf{A}}\right)\times\vec{\mathbf{A}} + \tfrac{1}{2}\vec{\nabla}\left(\vec{\mathbf{A}}\bullet\vec{\mathbf{A}}\right)$

$$\therefore \quad \left(\vec{\nabla}\times\vec{\mathbf{v}}\right)\times\vec{\mathbf{v}} + \tfrac{1}{2}\vec{\nabla}v^2 = -\frac{\vec{\nabla}p}{\rho} - \vec{\nabla}\phi$$

Take the dot product of $\vec{\mathbf{v}}$ on both sides ...

Equation becomes $\vec{\mathbf{v}}\bullet\vec{\nabla}\left\{\dfrac{p}{\rho} + \tfrac{1}{2}v^2 + \phi\right\} = 0$

since $\vec{\mathbf{v}}\bullet\left\{\left(\vec{\nabla}\times\vec{\mathbf{v}}\right)\times\vec{\mathbf{v}}\right\} = 0$

Bernoulli's theorem ... 2

This equation says that for a small displacement in the direction of the fluid velocity the quantity inside the curly brackets doesn't change.

We can write $\phi = gz$ for a gravitational field with the *z*-axis vertically upward.

In steady flow all displacements are along streamlines.
Then for all the points along a streamline, we can write:

$$\frac{p}{\rho} + \frac{v^2}{2} + gz = \text{constant (along a streamline)}$$

Bernoulli's theorem

This equation only holds for inviscid, steady, incompressible flow along a steamline.

Bernoulli's theorem ... 3

The constant may, in general, be different for different streamlines, but the left hand side of the equation is the same all along a given streamline.

It is often convenient to express the Bernoulli Equation between two points **along a streamline**:

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

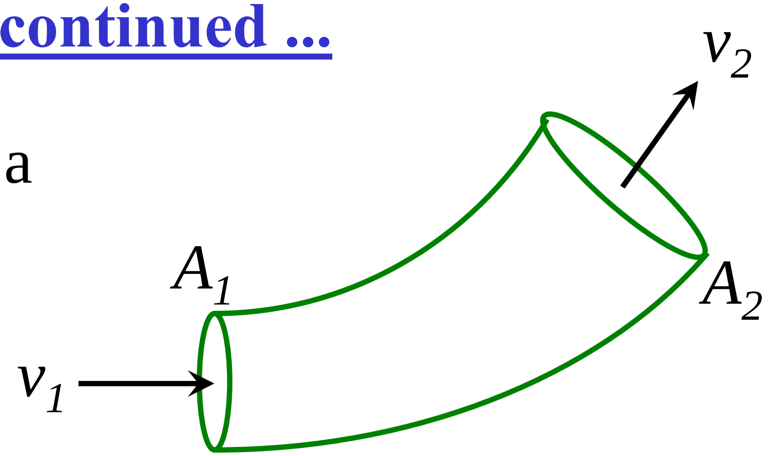
If the flow is also **irrotational** (see later), then

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

for <u>any</u> two points in the flow field.

Flow of dry water continued ...

Consider a fluid flowing through a streamtube as shown.



Equation of continuity:

$$\vec{\nabla} \bullet \vec{\mathbf{v}} = 0$$ for steady flow.

$$\therefore \int_V \vec{\nabla} \bullet \vec{\mathbf{v}} \, dV = 0$$ where V is the volume of the tube

Using the divergence theorem:

$$\int_V \vec{\nabla} \bullet \vec{\mathbf{v}} \, dV = \int_S \vec{\mathbf{v}} \bullet d\vec{\mathbf{a}} = 0$$

$$\therefore \int_S \vec{\mathbf{v}} \bullet d\vec{\mathbf{a}} = \int_{A_1} \vec{\mathbf{v}} \bullet d\vec{\mathbf{a}} + \int_{A_2} \vec{\mathbf{v}} \bullet d\vec{\mathbf{a}} + \overbrace{\int_{\text{outer walls}} \vec{\mathbf{v}} \bullet d\vec{\mathbf{a}}}^{=0}$$

$= 0$ ← no flux though outer walls

... flow of dry water continued ...

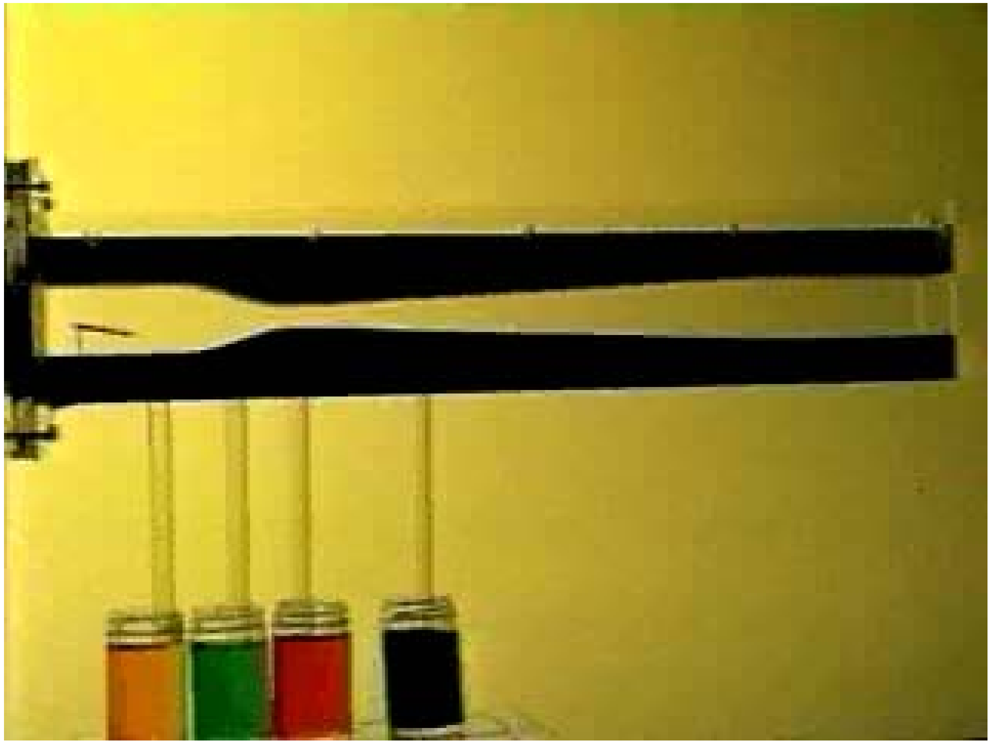
$$\therefore \int_{A_1} \vec{\mathbf{v}} \bullet d\vec{\mathbf{a}} = -\int_{A_2} \vec{\mathbf{v}} \bullet d\vec{\mathbf{a}}$$

If $\vec{\mathbf{v}}$ is parallel to ends of tube and the end surfaces are perpendicular to $\vec{\mathbf{v}}$, then

$$\vec{\mathbf{v}}_1 \bullet \vec{\mathbf{A}}_1 = -\vec{\mathbf{v}}_2 \bullet \vec{\mathbf{A}}_2$$

or $$v_1 A_1 = v_2 A_2$$

$\therefore$ if the streamtube narrows, the fluid speed v must increase to compensate.



<u>Video clip (3.6) A Venturi Channel:</u> As a fluid flows through a converging channel (Venturi channel), the pressure is reduced in accordance with the continuity and Bernoulli equations. As predicted by the continuity equation, the velocity of air flowing through the channel increases due to the reduction in the channel area. As predicted by the Bernoulli equation, an increase in velocity will cause a decrease in pressure. The attached water columns show that the greatest pressure reduction occurs at the narrowest part of the channel. The same principle is used in a garden sprayer so that liquid chemicals can be sucked from the bottle and mixed with water in the hose.

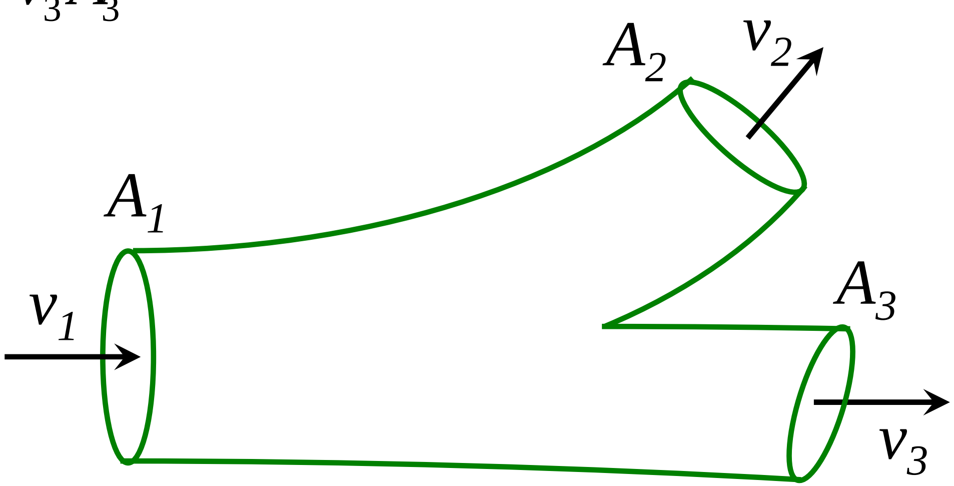
... flow of dry water continued ...

The continuity equation can be applied to determine the relation between the flows into and out of a **junction**.

For steady conditions:

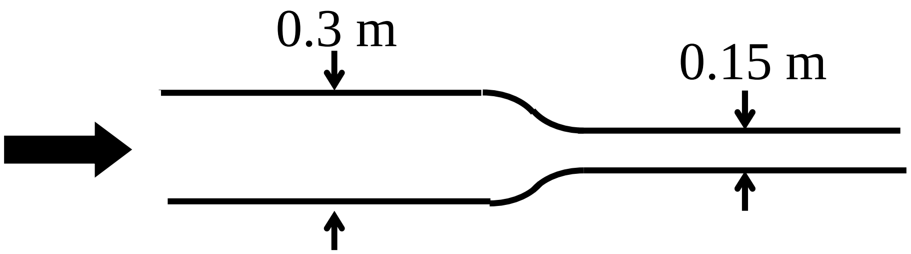
total inflow to junction = total outflow from junction

or $$v_1 A_1 = v_2 A_2 + v_3 A_3$$



Example:

A fluid flows from a 0.3 m diameter pipe in which the pressure is 300 kPa into a 0.15 m diameter pipe in which the pressure is 120 kPa. If the pipes are horizontal and viscous effects are negligible, determine the flowrate.



D_2

<u>Video clip (10.2) Merging Channels:</u> The simplest type of open channel flow is one for which the channel cross-sectional size and shape and the water depth remain constant along the length of the channel. In many situations these conditions are not met. Although the flow in each of the two merging channels of the model study shown may be essentially uniform flow upstream of their confluence, the actual merging of the two streams is quite complex and far from uniform flow. How the two streams mix to produce a single stream may be of considerable importance to downstream locations.

Flux

The "transport" of fluid properties by the flow of a fluid across a surface is embodied across a surface per unit time

The flux of (something) is the amount of that (something) being transported across a surface per unit time.

There are a number of fluxes that can be considered.

For a closed surface of volume V :

$$\textbf{Volume flux} = \int_S \vec{\mathbf{v}} \bullet d\vec{\mathbf{a}} = \int_S \vec{\mathbf{v}} \bullet \hat{\mathbf{n}}\, da$$

where $d\vec{\mathbf{a}} = \hat{\mathbf{n}}\, da$, where $\hat{\mathbf{n}}$ is a unit vector pointing outwards at right angles to S.

units of volume flux : $m^3\ s^{-1}$

Flux ... 2

Other types of flux can be defined:

Mass flux $= \int_S \rho \vec{\mathbf{v}} \bullet d\vec{\mathbf{a}} = \rho \int_S \vec{\mathbf{v}} \bullet \hat{\mathbf{n}}\, da$

units of mass flux : kg s^{-1}

Momentum flux $= \int_S (\rho \vec{\mathbf{v}}) \vec{\mathbf{v}} \bullet d\vec{\mathbf{a}} = \rho \int_S \vec{\mathbf{v}} (\vec{\mathbf{v}} \bullet \hat{\mathbf{n}})\, da$

units of momentum flux : kg m s^{-1} s^{-1} = N

Kinetic energy flux $= \int_S \left(\tfrac{1}{2} \rho v^2\right) \vec{\mathbf{v}} \bullet d\vec{\mathbf{a}} = \tfrac{1}{2} \rho v^2 \int_S \vec{\mathbf{v}} \bullet \hat{\mathbf{n}}\, da$

units of energy flux : J s^{-1} = W

Vorticity and rotation

Back to $\rho \dfrac{D\vec{\mathbf{v}}}{Dt} = -\vec{\nabla} p - \rho \vec{\nabla} \phi$

Use the identity: $\left(\vec{\mathbf{A}} \bullet \vec{\nabla}\right)\vec{\mathbf{A}} = \left(\vec{\nabla} \times \vec{\mathbf{A}}\right) \times \vec{\mathbf{A}} + \tfrac{1}{2}\vec{\nabla}\left(\vec{\mathbf{A}} \bullet \vec{\mathbf{A}}\right)$

and introduce a new vector field $\vec{\Omega}$ as curl $\vec{\mathbf{v}}$: $\vec{\Omega} = \vec{\nabla} \times \vec{\mathbf{v}}$

$$\therefore \left(\vec{\mathbf{v}} \bullet \vec{\nabla}\right)\vec{\mathbf{v}} = \vec{\Omega} \times \vec{\mathbf{v}} + \tfrac{1}{2}\vec{\nabla} v^2$$

and our equation of motion becomes:

$$\frac{\partial \vec{\mathbf{v}}}{\partial t} + \vec{\Omega} \times \vec{\mathbf{v}} + \tfrac{1}{2}\vec{\nabla} v^2 = -\frac{\vec{\nabla} p}{\rho} - \vec{\nabla} \phi$$

The vector field $\vec{\Omega}$ is called the **vorticity**.

Recall the circulation of a vector field. The circulation around any closed loop in a fluid is the line integral of the fluid velocity at a given instant in time, around the loop.

In other words, circulation = $\oint \vec{\mathbf{v}} \bullet d\vec{l}$

By Stokes' theorem: $\oint \vec{\mathbf{v}} \bullet d\vec{l} = \int_S \left(\vec{\nabla} \times \vec{\mathbf{v}}\right) \bullet d\vec{a}$

Then the circulation = $\int_S \vec{\Omega} \bullet d\vec{a}$

and the vorticity $\vec{\Omega}$ is the circulation around a unit area of fluid (perpendicular to the direction of $\vec{\Omega}$).

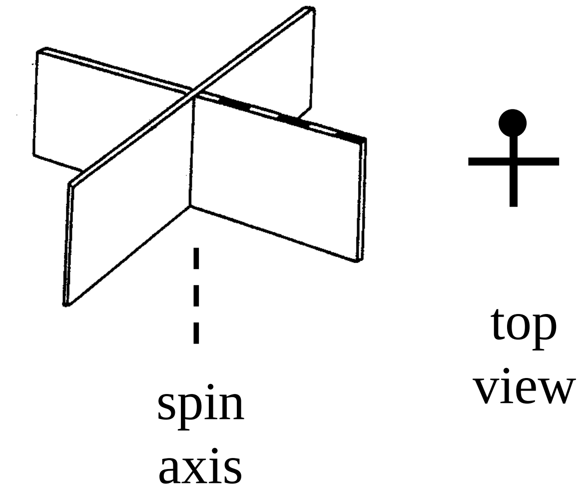
Vorticity

Define **vorticity**:

$$\vec{\Omega} = \vec{\nabla} \times \vec{\mathbf{v}}$$

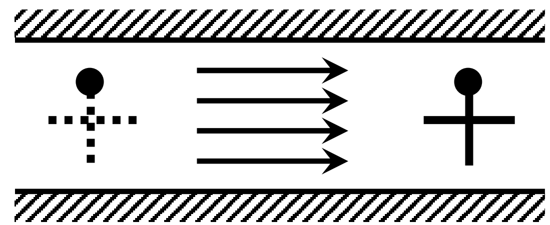
unit: s^{-1}

Introduce a **vorticity meter**:

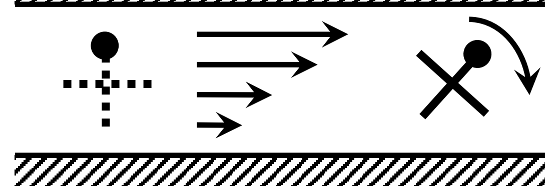


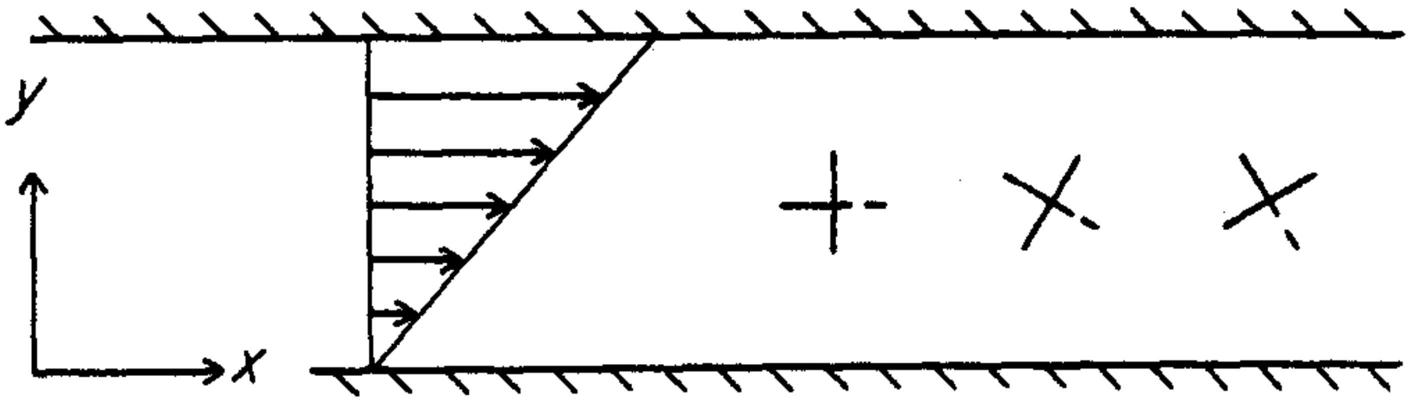
If the meter (paddle wheel) rotates in a moving fluid
$\rightarrow$ indicates the presence of vorticity

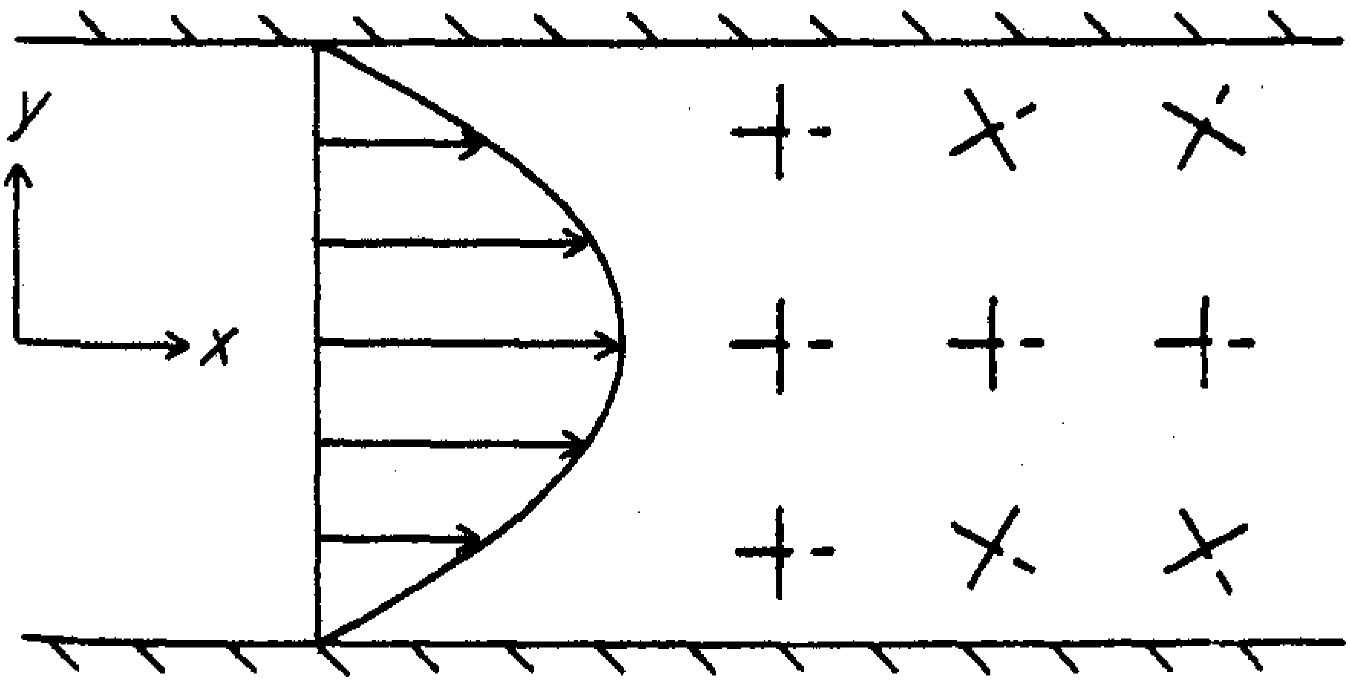
If the motion is in the straight line and the velocity is constant across the fluid, as shown, then the vorticity meter will <u>not rotate</u>.



If the velocity varies across the parallel flow, the vorticity meter will <u>rotate</u> in clockwise sense as it drifts.




y
x


y
x

Vorticity … 2

If $\vec{\Omega} = 0$ (everywhere), then the flow is called **irrotational**

If $\vec{\Omega} \neq 0$ (somewhere), then the flow is called **rotational**

$$\text{Circulation} = \oint \vec{\mathbf{v}} \bullet d\vec{l} = \int_S \Omega \bullet d\vec{a}$$

Think about this: a little bit of dirt (not an infinitesimal point) at any position in the field will rotate with angular velocity $\vec{\Omega}/2$

Can you prove this?

Example:

For a certain steady, two-dimensional flow field, the velocity is described by:

$$\vec{\mathbf{v}} = 4xy\hat{\mathbf{i}} + 2(x^2 - y^2)\hat{\mathbf{j}}$$

Is the flow rotational?

Summary so far ...

Equations of motion for the flow of dry water:

$$\vec{\nabla} \bullet \vec{\mathbf{v}} = 0 \qquad \text{(conservation of matter)}$$

$$\vec{\Omega} = \vec{\nabla} \times \vec{\mathbf{v}} \qquad \text{(definition of vorticity)}$$

$$\frac{\partial \vec{\mathbf{v}}}{\partial t} + \vec{\Omega} \times \vec{\mathbf{v}} + \frac{\vec{\nabla} v^2}{2} = -\frac{\vec{\nabla} p}{\rho} - \vec{\nabla} \phi \qquad \text{(Newton II)}$$

If we are only interested in the velocity field, then we can eliminate the pressure by

$$\vec{\nabla} \times \left(\frac{\partial \vec{\mathbf{v}}}{\partial t} + \vec{\Omega} \times \vec{\mathbf{v}} + \frac{\vec{\nabla} v^2}{2} \right) = \vec{\nabla} \times \left(-\frac{\vec{\nabla} p}{\rho} - \vec{\nabla} \phi \right)$$

➡ $$\frac{\partial \vec{\Omega}}{\partial t} + \vec{\nabla} \times \left(\vec{\Omega} \times \vec{\mathbf{v}} \right) = 0 \qquad \{\text{since curl (grad } \phi) = 0\}$$

We can now specify the velocity field $\vec{\mathbf{v}}$ at any time …

If we know $\vec{\Omega}$ at time t

from $\vec{\Omega} = \vec{\nabla} \times \vec{\mathbf{v}}$ and $\vec{\nabla} \bullet \vec{\mathbf{v}} = 0$

we know $\vec{\mathbf{v}}$ everywhere

from $$\frac{\partial \vec{\Omega}}{\partial t} = -\vec{\nabla} \times \left(\vec{\Omega} \times \vec{\mathbf{v}} \right)$$

we know $$\frac{\partial \vec{\Omega}}{\partial t}$$

and hence $\vec{\Omega}$ at $t' > t$

and cycle around again

If $\vec{\Omega} = 0$ everywhere at any time t ,

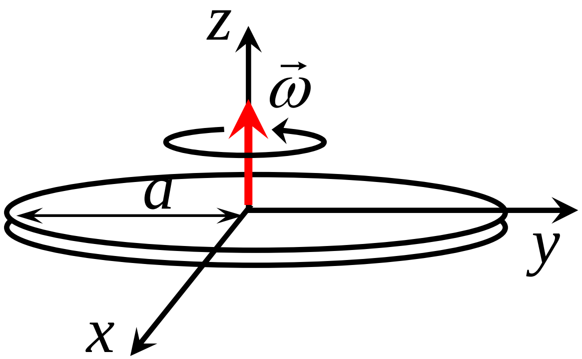
then $\dfrac{\partial \vec{\Omega}}{\partial t} = 0$ everywhere and $\vec{\Omega} = 0$ at $t + \Delta t$.

If the flow started as irrotational, then it is permanently irrotational.

We only need to solve $\vec{\nabla} \bullet \vec{\mathbf{v}} = 0$ and $\vec{\nabla} \times \vec{\mathbf{v}} = 0$.

(These velocity fields look like electrostatic or magnetostatic fields in free space.)

Example: The rotating fluid disc



Consider a fluid disc of radius a rotating like a rigid body at an angular velocity $\vec{\omega}$.

$$\text{Vorticity} = \vec{\nabla}\times\vec{\mathbf{v}} = \vec{\nabla}\times\left(\vec{\omega}\times\vec{\mathbf{r}}\right)$$

$$= \vec{\nabla}\times\left\{\left(0\hat{\mathbf{i}} + 0\hat{\mathbf{j}} + \omega\hat{\mathbf{k}}\right)\times\left(x\hat{\mathbf{i}} + y\hat{\mathbf{j}} + 0\hat{\mathbf{k}}\right)\right\}$$

$$= \vec{\nabla}\times\left\{-y\omega\hat{\mathbf{i}} + x\omega\hat{\mathbf{j}}\right\} = 2\omega\hat{\mathbf{k}}$$

$$\text{Circulation, } \Gamma = \oint \vec{\mathbf{v}}\bullet d\vec{l} = \oint \omega a\, dl = \omega a\left(2\pi a\right)$$

$$= 2\omega\left(\pi a^2\right) = 2\omega A$$

area of the disc

The circulation = vorticity * disc area
(true for all rigid body rotation)

Vorticity is a microscopic, vector, point-wise measure of fluid element rotation.
Circulation is a macroscopic, scalar, area-wise measure of fluid rotation.

Vorticity is the measure of the fluid rotational aspects of an infinitesimal fluid parcel.
Circulation includes the fluid rotational tendencies over a finite area of fluid.

The velocity potential

The condition of irrotational flow $\vec{\nabla} \times \vec{\mathbf{v}} = 0$ means that, in Cartesian coordinates:

$$\frac{\partial v_x}{\partial y} = \frac{\partial v_y}{\partial x} \;,\; \frac{\partial v_y}{\partial z} = \frac{\partial v_z}{\partial y} \quad \text{and} \quad \frac{\partial v_z}{\partial x} = \frac{\partial v_x}{\partial z} \quad .$$

It is possible to satisfy these equations by defining a **potential function** ϕ such that

$$v_x = \frac{\partial \phi}{\partial x} \;,\; v_y = \frac{\partial \phi}{\partial y} \quad \text{and} \quad v_z = \frac{\partial \phi}{\partial z} \quad .$$

so that

$$\frac{\partial v_x}{\partial y} = \frac{\partial^2 \phi}{\partial x \partial y} \;,\; \frac{\partial v_y}{\partial x} = \frac{\partial^2 \phi}{\partial y \partial x} \quad \text{... etc}$$

In terms of ϕ the velocity may be written as

$$\vec{\mathbf{v}} = v_x \hat{\mathbf{i}} + v_y \hat{\mathbf{j}} + v_z \hat{\mathbf{k}} = \frac{\partial \phi}{\partial x}\hat{\mathbf{i}} + \frac{\partial \phi}{\partial y}\hat{\mathbf{j}} + \frac{\partial \phi}{\partial z}\hat{\mathbf{k}} \quad \text{or} \quad \vec{\mathbf{v}} = \vec{\nabla}\phi$$

The velocity potential ... 2

For an irrotational flow: $\vec{\mathbf{v}} = \vec{\nabla}\phi$

... the velocity is expressible as the **gradient** of a scalar function ϕ.

The velocity potential is a consequence of the **irrotationality** of the flow field, whereas the stream function is a consequence of conservation of mass.

It is to be noted, however, that the velocity potential can be defined for a general three-dimensional flow, whereas the stream function is restricted to two-dimensional flows.

For an incompressible fluid we know from conservation of mass that $\vec{\nabla} \bullet \vec{\mathbf{v}} = 0$.

and therefore for incompressible, irrotational flow (with $\vec{\mathbf{v}} = \vec{\nabla}\phi$) it follows that $\nabla^2 \phi = 0$,

where ∇^2 is the **Laplacian operator**.

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

This differential equation arises in many different areas of engineering and physics and is called **Laplace's equation**. Thus, inviscid, incompressible, irrotational flow fields are governed by Laplace's equation.
This type of flow is commonly called a **potential flow.**

Example:

A steady, incompressible, two-dimensional flow field is described by

$$\vec{\mathbf{v}} = (x - 4y)\hat{\mathbf{i}} + (-y - 4x)\hat{\mathbf{j}}$$

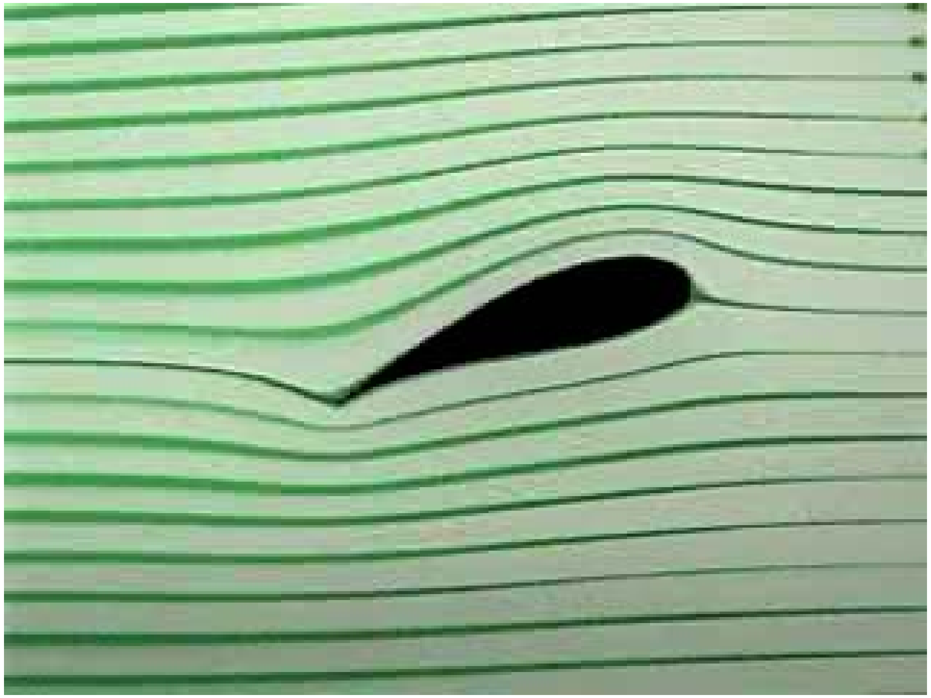
Show that the flow satisfies the continuity equation and obtain the expression for the steam function. If the flow is potential, then obtain the expression for the velocity potential.

Example:

Show that $\phi = x^3 - 3xy^2$ is a valid velocity potential and that it describes an incompressible flow field.

Determine the corresponding stream function.

Find the stagnation points, and the pressure distribution.



Video clip (6.4) Potential Flow: Flow fields for which an incompressible fluid is assumed to be frictionless and the motion to be irrotational are commonly referred to as potential flows. Paradoxically, potential flows can be simulated by a slowly moving, viscous flow between closely spaced parallel plates. For such a system, dye injected upstream reveals an approximate potential flow pattern around a streamlined airfoil shape. Similarly, the potential flow pattern around a bluff body is shown. Even at the rear of the bluff body the streamlines closely follow the body shape. Generally, however, the flow would separate at the rear of the body, an important phenomenon not accounted for with potential theory.

Examples of basic, plane potential flows in 2D

A major advantage of Laplace's equation is that it is a linear partial differential equation. Since it is linear, various solutions can be added to obtain other solutions.

For example, if ϕ_1 and ϕ_2 are two solutions to Laplace's equation, then $\phi_1 + \phi_2$ is also a solution.

The practical implication of this result is that if we have certain basic solutions we can combine them to obtain more complicated and interesting solutions.

For 2D flow: $v_x = \dfrac{\partial \phi}{\partial x} \quad v_y = \dfrac{\partial \phi}{\partial y}$ and $v_x = \dfrac{\partial \psi}{\partial y} \quad v_y = -\dfrac{\partial \psi}{\partial x}$

where ψ is the stream function.

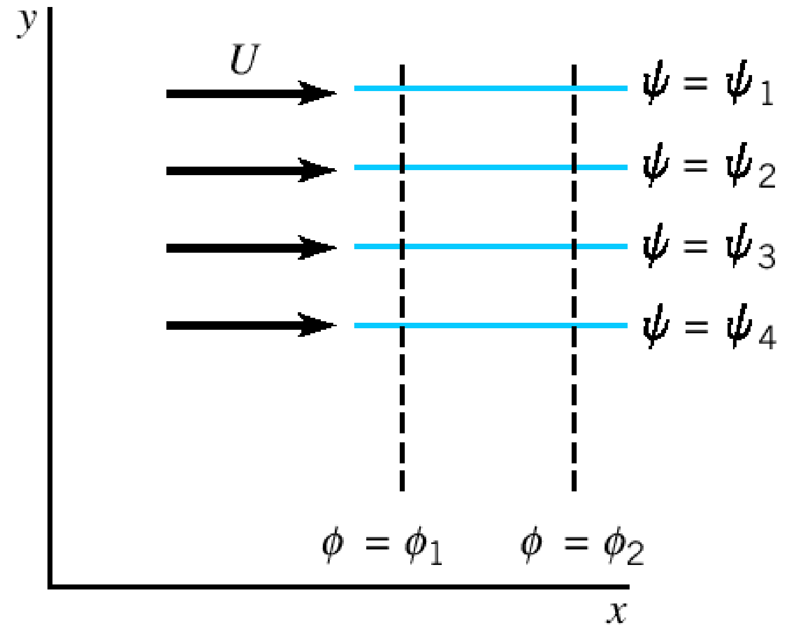
In cylindrical coordinates:

$$v_r = \frac{\partial \phi}{\partial r} \qquad v_\theta = \frac{1}{r}\frac{\partial \phi}{\partial \theta} \qquad \text{and} \quad v_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} \qquad v_\theta = -\frac{\partial \psi}{\partial r}$$

Uniform flow

Simplest plane flow, since all streamlines straight and parallel
Magnitude of velocity is constant
(in the *x*-direction)

In this case: $\frac{\partial \phi}{\partial x} = U \qquad \frac{\partial \phi}{\partial y} = 0$



Integrating yields $\phi = Ux + C$

where *C* is an arbitrary constant which is set to 0.

Then $\phi = Ux$

The stream function can be obtained in a similar way, since:

$$\frac{\partial \psi}{\partial y} = U \quad \text{and} \quad \frac{\partial \psi}{\partial x} = 0 \qquad \text{therefore} \quad \psi = Uy$$

Uniform flow ... 2

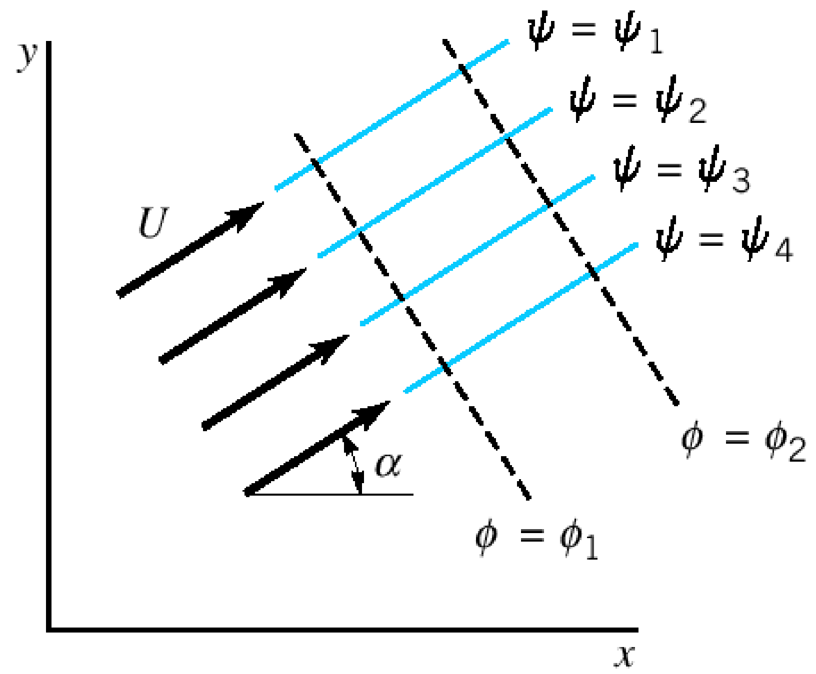
These results can be generalized to provide the velocity potential and stream function for a uniform flow at an angle α with the x-axis.

For this case

$$\phi = U\left(x\cos\alpha + y\sin\alpha\right)$$

and

$$\psi = U\left(y\cos\alpha - x\sin\alpha\right)$$



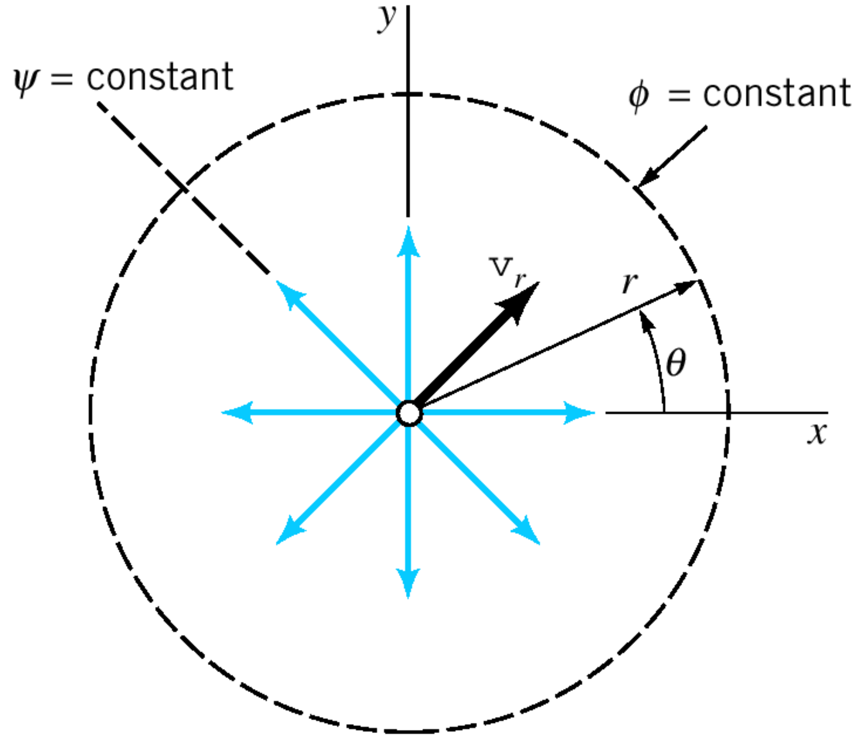
Source and sink

Consider a fluid flowing radially outward from a line through the origin perpendicular to the *x*–*y* plane as shown.

Let *m* be the volume rate of flow emanating from the line (per unit length).

To satisfy conservation of mass:

$$(2\pi r) v_r = m$$



Also, since the flow is a purely radial, the corresponding velocity potential can be obtained by integrating the equations

$$\frac{\partial \phi}{\partial r} = \frac{m}{2\pi r} \qquad \frac{1}{r}\frac{\partial \phi}{\partial \theta} = 0$$

giving $\phi = \dfrac{m}{2\pi} \ln r$

Source and sink ... 2

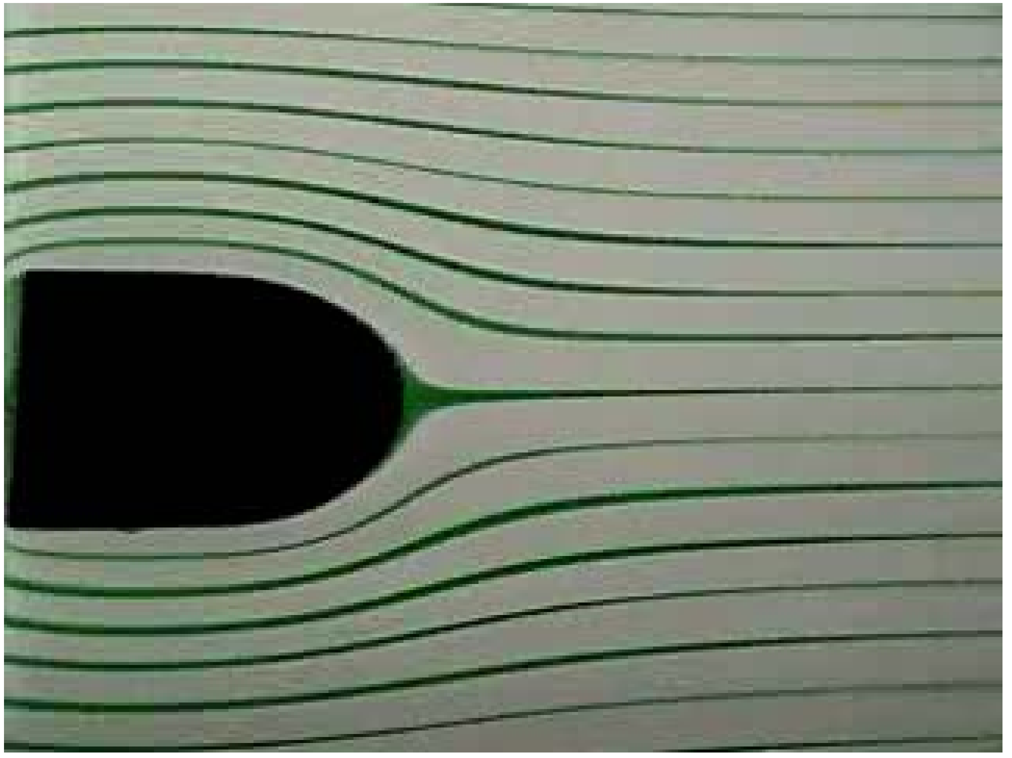
If m is positive, the flow is radially outward, and the flow is considered to be a *source* flow.
If m is negative, the flow is toward the origin, and the flow is considered to be a *sink* flow.
The flowrate, m is the *strength* of the source or sink.

The stream function for the source can be obtained by integrating

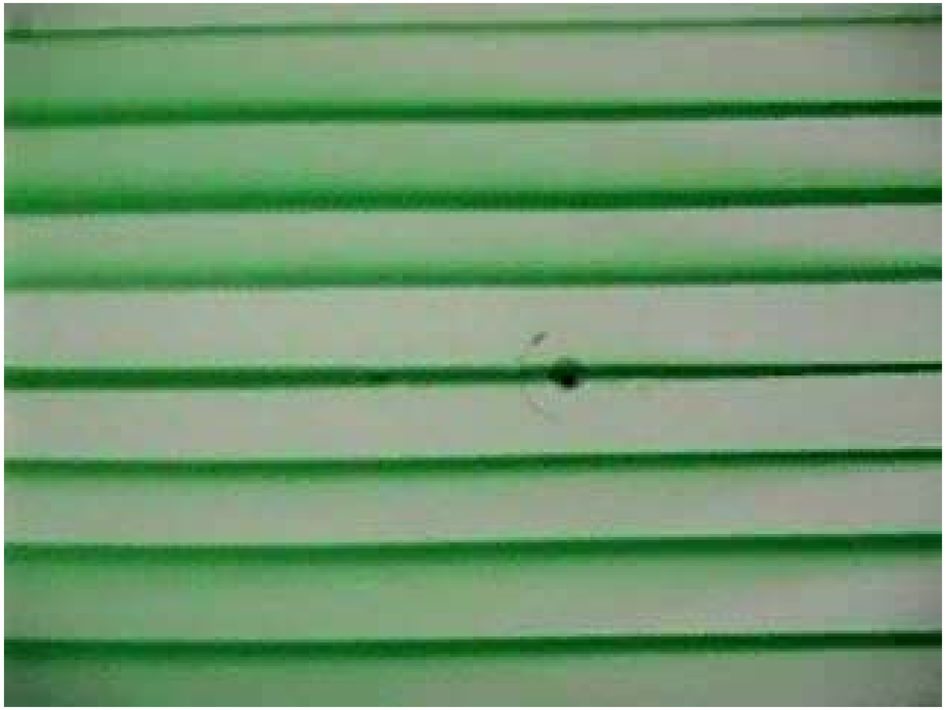
$$\frac{\partial \psi}{\partial r} = 0 \qquad \frac{1}{r}\frac{\partial \psi}{\partial \theta} = \frac{m}{2\pi r}$$

giving $$\psi = \frac{m}{2\pi}\theta$$

It is apparent that the streamlines lines are radial lines, and the equipotential lines are concentric circles centered at the origin.



Video clip (3.3) Stagnation Point Flow: On any body in a flowing fluid there is a stagnation point. Some of the fluid flows “over” and some “under” the body. The dividing line (the stagnation streamline) terminates at the stagnation point on the body. As indicated by the dye filaments in the water flowing past a streamlined object, the velocity decreases as the fluid approaches the stagnation point. The pressure at the stagnation point (the stagnation pressure) is that pressure obtained when a flowing fluid is decelerated to zero speed by a frictionless process.



Video clip (6.3) Half-Body: Basic velocity potentials and stream functions can be combined to describe potential flow around various body shapes. The combination of a uniform flow and a source can be used to describe the flow around a streamlined body placed in a uniform stream. Streamlines created by injecting dye in steadily flowing water show a uniform flow. Source flow is created by injecting water through a small hole. It is observed that for this combination the streamline passing through the stagnation point could be replaced by a solid boundary which resembles a streamlined body in a uniform flow. The body is open at the downstream end and is thus called a half-body.

Potential vortex flow

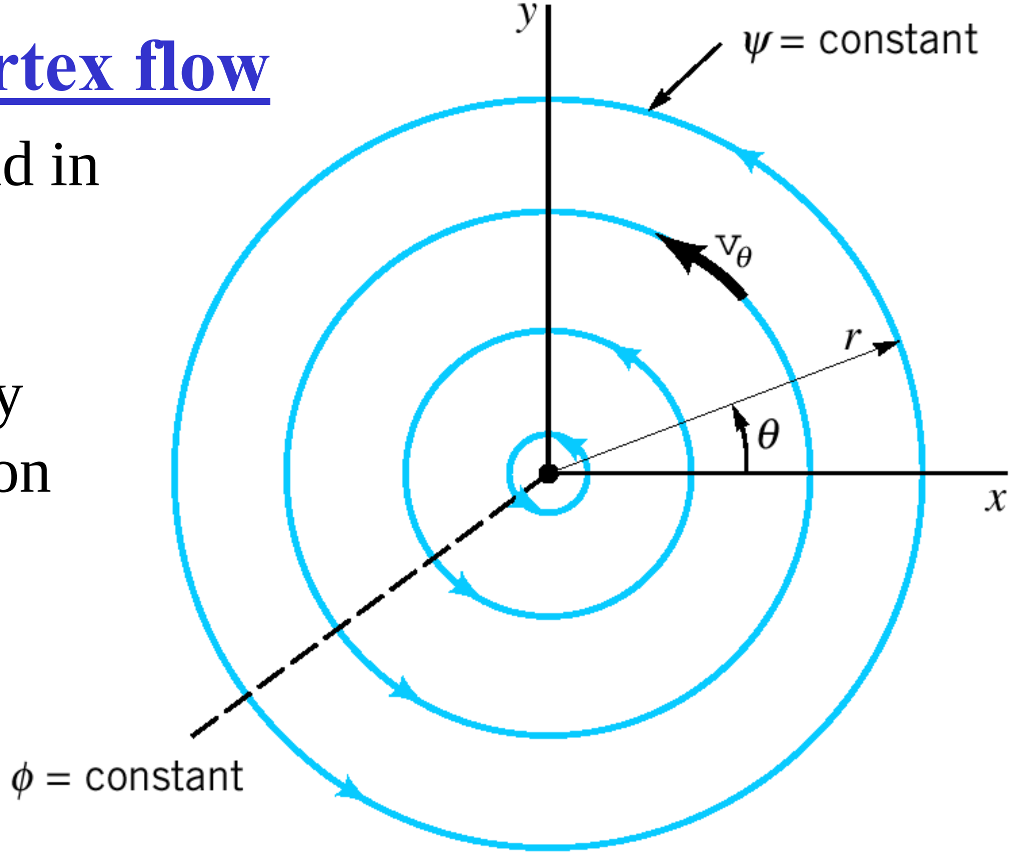
W next consider a flow field in which the streamlines are concentric circles.

We interchange the velocity potential and stream function for the source.

Thus, let $\phi = K\theta$

and $\psi = -K \ln r$

where K is a constant.



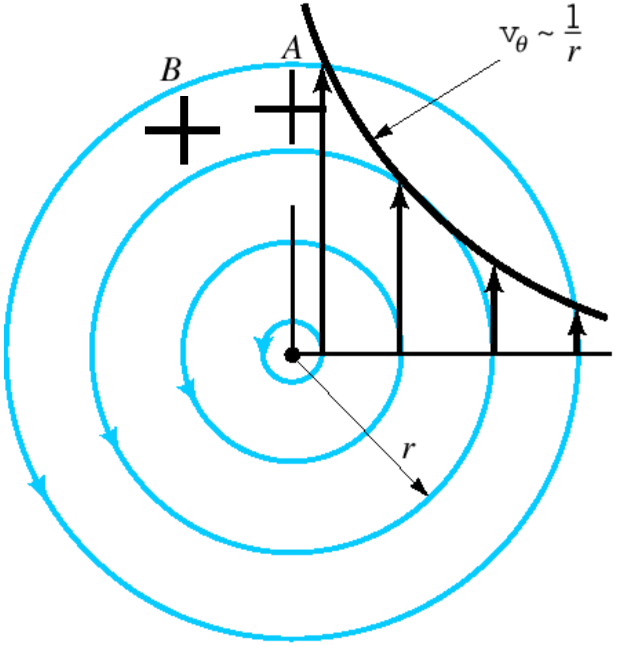
In this case the streamlines are concentric circles with $v_r = 0$ and

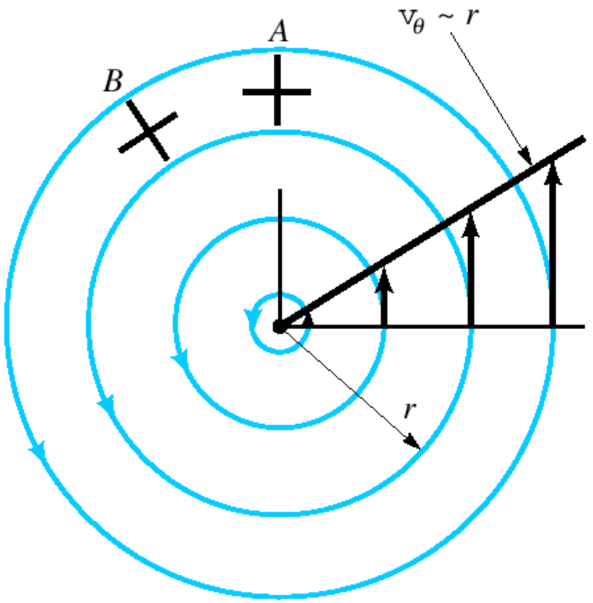
$$v_\theta = \frac{1}{r}\frac{\partial \phi}{\partial \theta} = -\frac{\partial \psi}{\partial r} = \frac{K}{r}$$

This result indicates that the tangential velocity varies inversely with the distance from the origin, with a singularity occurring at the origin where the velocity becomes infinite.

Potential vortex flow ... 2

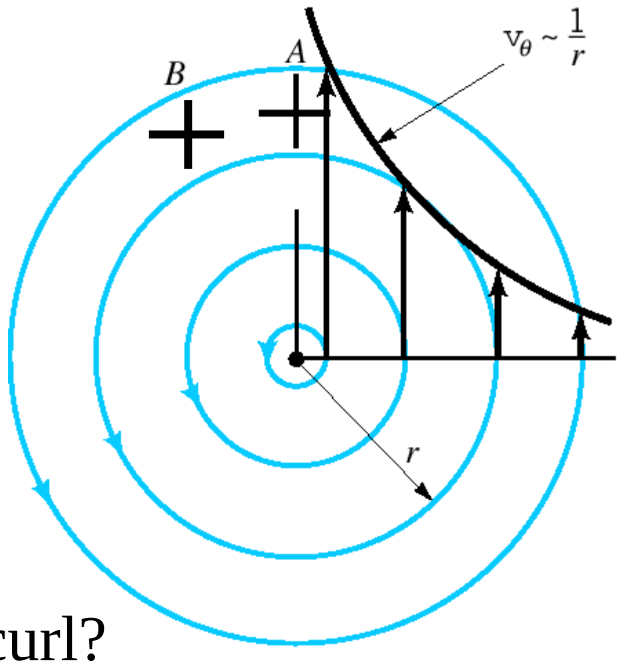
It may seem strange that this *vortex* motion is irrotational (and it is since the flow field is described by a velocity potential)
However, it must be recalled that rotation refers to the orientation of a fluid element and not the path followed by the element. Thus, for an irrotational vortex, if a vorticity meter was placed in the flow field at location A, as indicated, then the meter would not rotate as it moved to location B.





If the fluid were rotating as a rigid body, such that where $v_\theta = Kr$, then a vorticity meter similarly placed in the flow field would rotate as shown. This type of vortex motion is **rotational** and cannot be described with a velocity potential. The rotational vortex is commonly called a **forced vortex** whereas the irrotational vortex is usually called a **free vortex**.

Potential vortex flow ... 3



Recall the circulation: $\Gamma = \oint_C \vec{\mathbf{v}} \bullet d\vec{l}$

For our free vortex, clearly the liquid is circulating

But $\vec{\nabla} \times \vec{\mathbf{v}} = 0$ in the liquid

How can there be a circulation without a curl?

$\rightarrow$ the line integral of $\vec{\mathbf{v}}$ around any loop enclosing the cylinder is not zero, although the line integral of $\vec{\mathbf{v}}$ around any closed path which does not include the cylinder is zero

$$\Gamma = \int_0^{2\pi} \frac{K}{r} r d\theta = 2\pi K$$

Potential vortex flow ... 4

The velocity potential and stream function for the **free vortex** are commonly expressed in terms of the circulation as

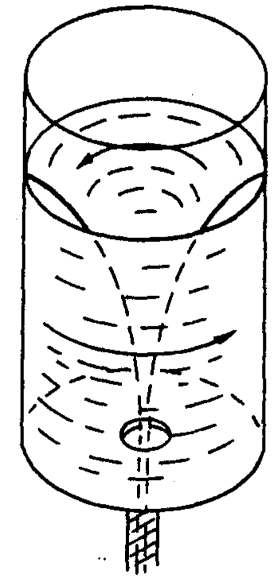
$$\phi = \frac{\Gamma}{2\pi}\theta$$

and

$$\psi = -\frac{\Gamma}{2\pi}\ln r$$

The velocity field of a fluid in an irrotational circulation around a cylinder is analogous to the magnetic field circulating around a current-carrying wire.

Think of the water moving down the drain in your bath…
As a "particle" of water moves inwards it picks up speed by the conservation of an angular momentum, and the water travels radially towards the hole.



From $\vec{\nabla} \bullet \vec{\mathbf{v}} = 0$ it follows that the radial velocity $\propto 1/r$
So the total velocity increases as $1/r$ and the water "particles" spiral inwards towards the hole.

The air water surface is at atmospheric pressure, so that

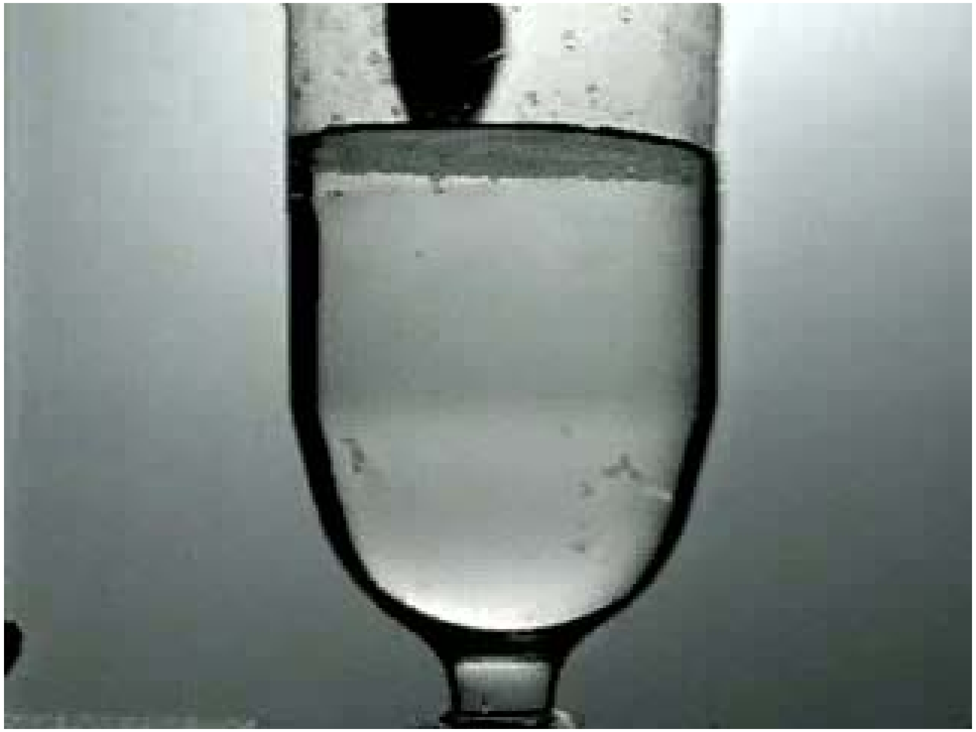
$$gz + \tfrac{1}{2} m v^2 = \text{constant}$$

but $v \propto \dfrac{1}{r}$ so the shape of the vortex surface is $\left(z - z_0\right) = \dfrac{k}{r^2}$

which describes the typical "whirlpool" shape.

A **combined vortex** is one with a forced vortex as a central core and a velocity distribution corresponding to that of a free vortex outside the core.

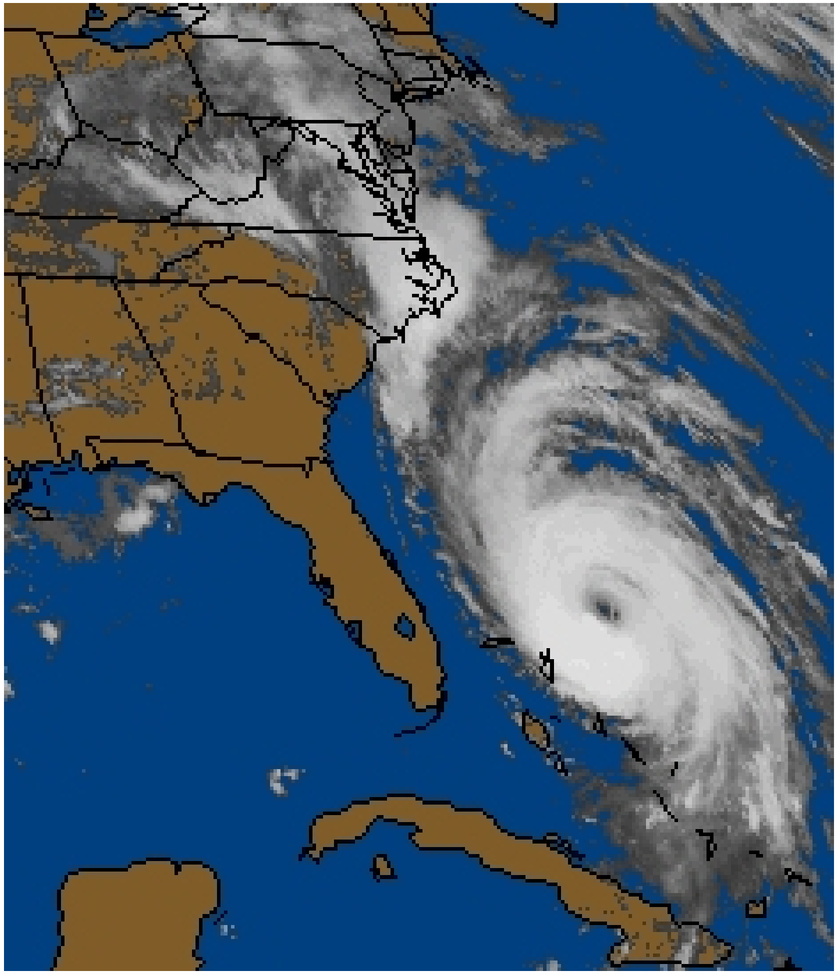
Whirlpools, tornadoes and water spouts may be described in this way. They are all examples of a free vortex since the fluid flows in nearly circular paths with nearly zero vorticity. Only near the centre of the free vortex does the vorticity deviate significantly from zero.



Video clip (3.2) Free Vortex

For flow with curved streamlines, centrifugal acceleration causes the pressure on the outside of the bend to be greater than on the inside. The swirling water draining from an inverted bottle approximates a free vortex. The velocity increase and pressure decrease near the center of the flow produce a hollow air core. Similar flows in the atmosphere can produce tornados. The rotation (swirl) of the parent cloud is enhanced as the air is drawn into the strong updraft at the tornado core.

<u>Video clip (6.2) Vortex in a beaker:</u> A flow field in which the streamlines are concentric circles is called a vortex. A vortex is easily created using a magnetic stirrer. As the stir bar is rotated at the bottom of a beaker containing water, the fluid particles follow concentric circular paths. A relatively high tangential velocity is created near the center which decreases to zero at the beaker wall. This velocity distribution is similar to that of a free vortex, and the observed surface profile can be approximated using the Bernoulli equation which relates velocity, pressure, and elevation.



Example: The idealized tornado

Assume the vertical component of the velocity is negligible.
The tornado has a core of radius r and rotates at a constant angular velocity ω.

Within the core: $v_c = \omega r \qquad (r < r_o)$

and outside the core: $v_o = k / r \qquad (r > r_o)$ - a "free vortex"

If $v = v_c$ at $r = r_o$, then $\omega r_o = \dfrac{k}{r_o} \rightarrow k = \omega r_o^2$

and $v_o = \dfrac{\omega r_o^2}{r}$

Vorticity $\vec{\Omega} = \vec{\nabla} \times \vec{\mathbf{v}} = \vec{\nabla} \times \left(\vec{\omega} \times \vec{\mathbf{r}} \right)$ inside the core

In our coordinate system: $\vec{\omega} = 0\hat{\mathbf{i}} + 0\hat{\mathbf{j}} + \omega\hat{\mathbf{k}}$

$$\vec{\mathbf{r}} = x\hat{\mathbf{i}} + y\hat{\mathbf{j}} + 0\hat{\mathbf{k}}$$

Then $\vec{\Omega} = \vec{\nabla} \times \left(\vec{\omega} \times \vec{\mathbf{r}} \right) = \vec{\nabla} \times \left(-y\omega\hat{\mathbf{i}} + x\omega\hat{\mathbf{j}} \right) = 2\omega\hat{\mathbf{k}}$ inside the core

Outside the core, $\vec{\Omega} = 0$ (free vortex)

The idealized tornado ... 2

What about the circulation ?

Inside the core:

$$\Gamma \;=\; \oint \vec{\mathbf{v}} \bullet d\vec{l} \;=\; \int_0^{2\pi} (\omega r) r\, d\theta$$

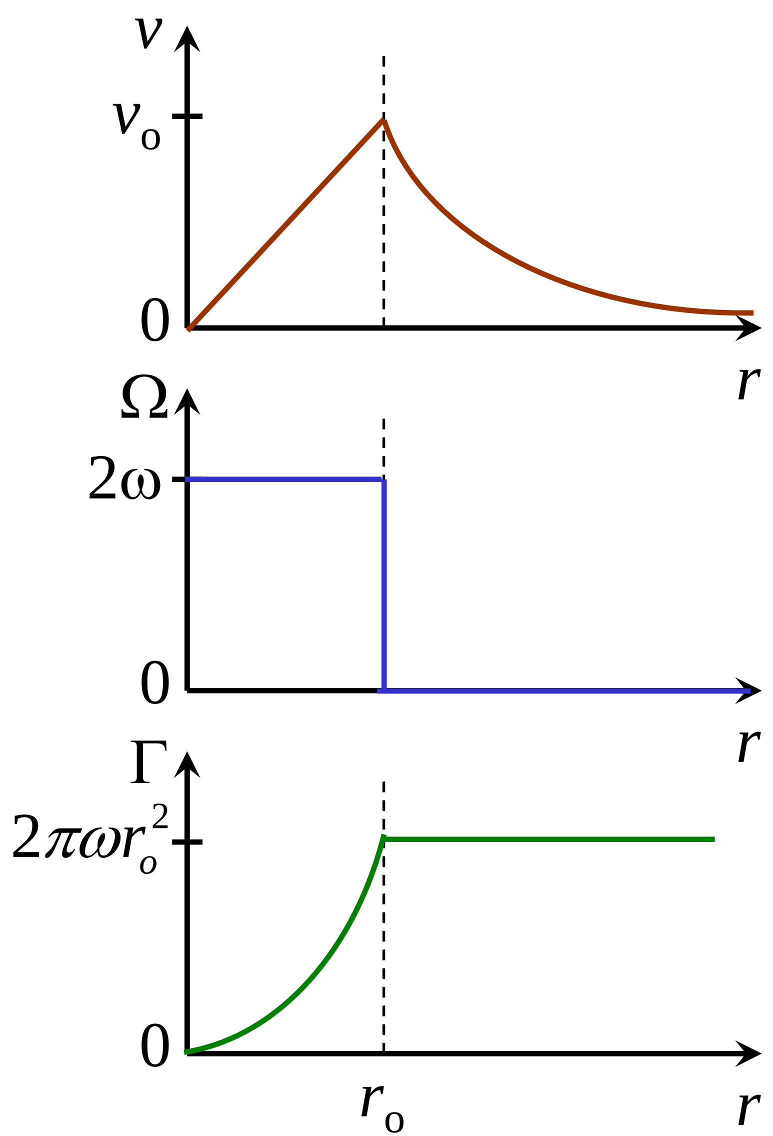
$$=\; 2\pi\omega r^2 \quad (r < r_o)$$

the circulation increases with *r* as expected, since circulation = (vorticity)(area)

Outside the core:

$$\Gamma \;=\; \oint \vec{\mathbf{v}} \bullet d\vec{l} \;=\; \int_0^{2\pi} \left(\frac{\omega r_o^2}{r} \right) r\, d\theta$$

$$=\; 2\pi\omega r_o^2 \quad (r > r_o)$$



Vortex lines

General equations for the flow of an **incompressible** fluid:

I $\vec{\nabla}\bullet\vec{\mathbf{v}} = 0$ II $\vec{\Omega} = \vec{\nabla}\times\vec{\mathbf{v}}$ III $\dfrac{\partial\vec{\Omega}}{\partial t} + \vec{\nabla}\times\left(\vec{\Omega}\times\vec{\mathbf{v}}\right) = 0$

Helmholtz describe the physics content the physics content of these theories in terms of 3 theorems……

Use **vortex lines -** field lines drawn in the direction of $\vec{\Omega}$ and have a density in any region which is proportional to the magnitude of $\vec{\Omega}$.

From equation II : $\nabla\bullet\vec{\Omega} = \nabla\bullet\left(\vec{\nabla}\times\vec{\mathbf{v}}\right) = 0$

i.e. the divergence of $\vec{\Omega} = 0$

Vortex lines always form closed loops (like $\vec{\mathbf{B}}$ lines)

Vortex lines ... 2

Equation III was also described by Helmholtz - vortex lines also move with the fluid.

If you colour the fluid particles along the vortex line at $t = 0$, then at $t = t'$, the particles mark the new position of the vortex line. Given the flow pattern at $t = 0$, the pattern can be determined at all t.

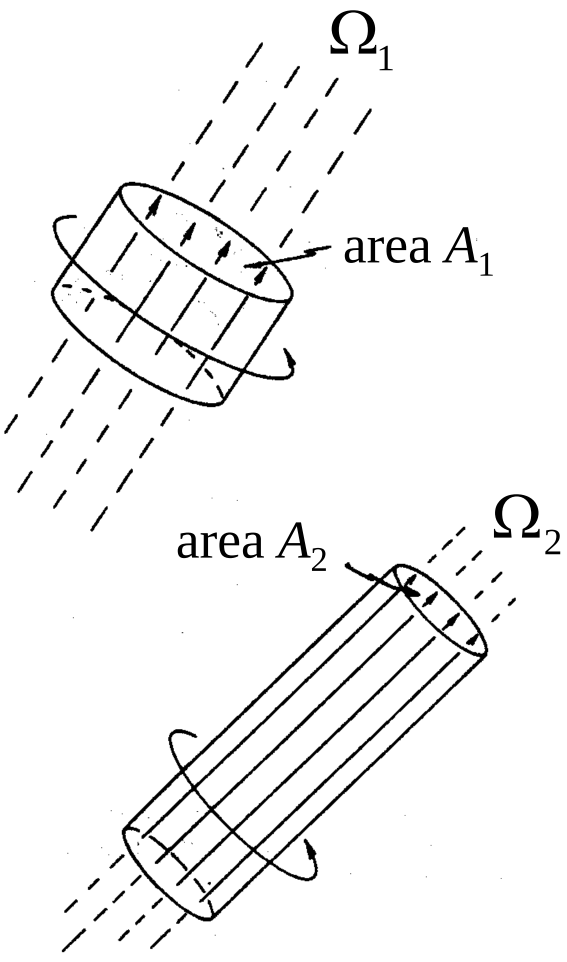
Given $\vec{\mathbf{v}} = 0$ everywhere $\rightarrow$ calculate $\vec{\Omega}$ $\left(= \vec{\nabla} \times \vec{\mathbf{v}}\right)$

Also, from $\vec{\mathbf{v}}$ you can tell where the vortex lines are going to be a little later (they move with speed v).
Then with the new $\vec{\Omega}$, use I & II to find the new $\vec{\mathbf{v}}$ and so on ...

If we have the flow pattern at one instant, it can in principle, be calculated at all subsequent times.

Vortex lines ... 3

Consider a small cylinder of liquid whose axis is parallel to the vortex lines. At some time later, the same piece of liquid will be somewhere else. Generally it will occupy a cylinder in a different position, having a different diameter and different orientation.



If the diameter of the cylinder has decreased, the length would have increased to keep the volume constant. Also, since the vortex lines are stuck with the material the density of the vortex lines increases as the diameter decreases.

According to Helmholtz: $\Omega_1 A_1 = \Omega_2 A_2$

Vortex lines ... 4

With zero viscosity, all the forces on the surface of the cylindrical volume are perpendicular to the surface.
The pressure forces can cause the volume to be moved from place to place, or change shape. But with no tangential forces, the angular momentum of the material cannot change.

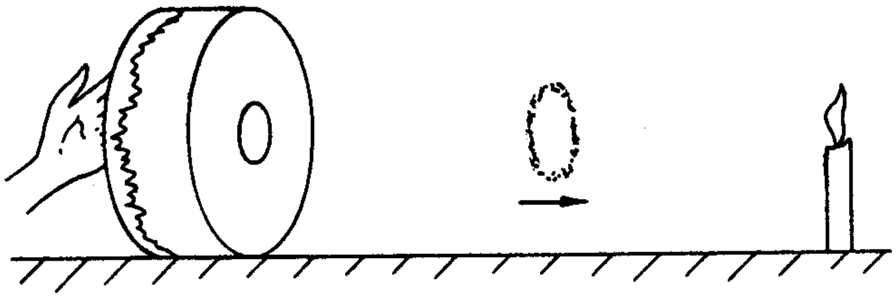
Since $I \propto mr^2$ for the cylinder, and $\omega \propto \Omega$

$$L_1 = L_2 \quad \rightarrow \; I_1\omega_1 = I_2\omega_2 \quad \rightarrow \; m_1 r_1^2 \Omega_1 = m_2 r_2^2 \Omega_2$$

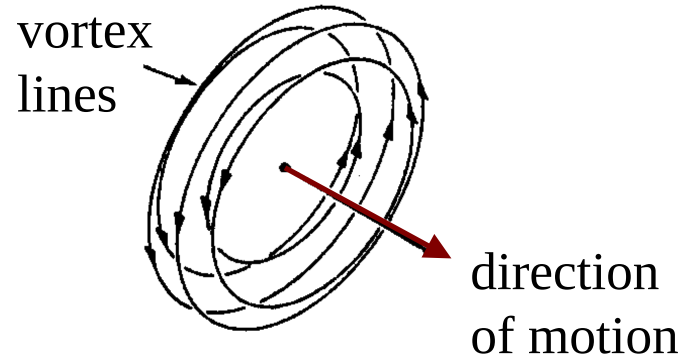
But $m_1 = m_2$ and $A \propto r^2 \Rightarrow \Omega_1 A_1 = \Omega_2 A_2$ again

In the absence of viscosity, the angular momentum of an element of the fluid cannot change.

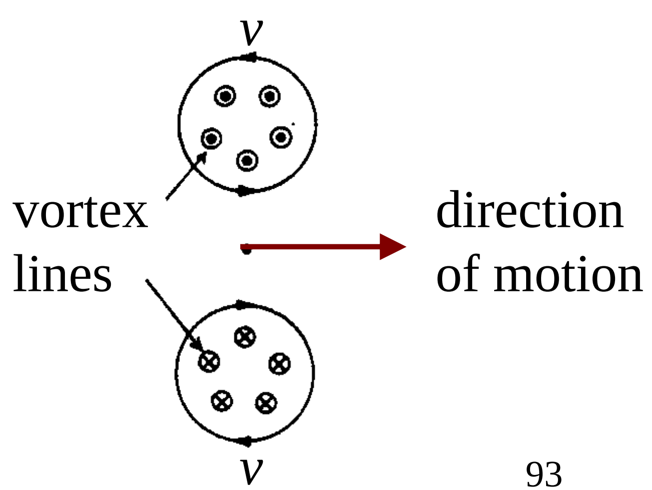
Moving vortices : smoke rings



a candle placed a few metres away is blown out by a smoke ring which travels at a finite speed



The smoke ring is torus-shaped bundle of vortex lines.
Since $\vec{\Omega} = \vec{\nabla} \times \vec{\mathbf{v}}$ these vortex lines also represents a circulation of $\vec{\mathbf{v}}$.



The circulating velocity around the bottom of the ring extends up to the top of the ring, having there a forward motion.

Since the lines of $\vec{\Omega}$ moving with the fluid, they also move ahead with the velocity $\vec{\mathbf{v}}$. Of course the circulation of $\vec{\mathbf{v}}$ around the top part of the ring is responsible for the forward motion of the vortex lines at the bottom.

But there is a problem here! $$\frac{\partial \vec{\Omega}}{\partial t} + \vec{\nabla} \times \left(\vec{\Omega} \times \vec{\mathbf{v}} \right) = 0$$

⇨ If $\vec{\Omega} = 0$ at $t = 0$, then $\vec{\Omega}$ will always be zero.

It is impossible to produce any vorticity under any circumstance ... but we have produced a vortex ring in it is impossible to produce any vorticity our drum, although $\vec{\mathbf{v}} = 0$, $\vec{\Omega} = 0$, everywhere in the drum before it was hit.

$\rightarrow$ There is a problem with our "dry" water model.

We need a theory of "wet" water to get a complete understanding of the behaviour of the fluid.

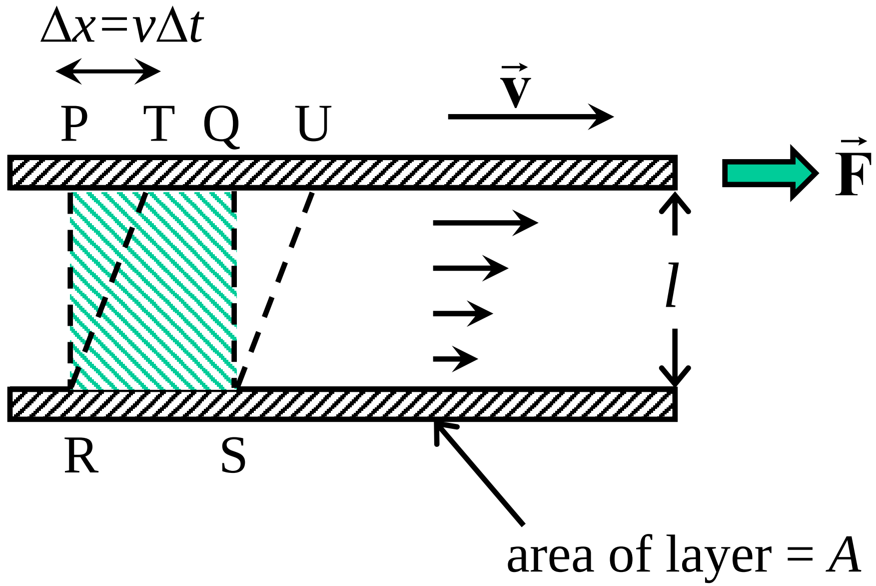
We need to include the forces arising from the **viscosity** of the fluid.

Viscosity

Although fluids do not support a shearing stress, they do offer some degree of resistance to shearing motion. This resistance to shearing motion is a form of internal friction, called **viscosity**, which arises as the adjacent layers of the fluid slide past one another.

Consider two parallel layers of fluid, one fixed and the other moving under the action of an external force $\vec{\mathbf{F}}$.

Adjacent layers of the fluid will be set into relative motion causing each portion (e.g. PQSR) to be distorted (into TUSR)



The fluid has undergone a shearing strain = $\dfrac{\Delta x}{l}$

Viscosity ... 2

The upper plate moves with speed v and the fluid adjacent to it moves at the same speed, then

The rate of change of shearing strain = $\frac{\text{strain}}{\Delta t} = \frac{\Delta x / l}{\Delta t} = \frac{v}{l}$

The coefficient of viscosity η for the fluid is defined as

$$\eta = \frac{\text{shearing stress}}{\text{rate of change of shearing strain}} = \frac{F / A}{v / l} = \frac{Fl}{Av}$$

SI units of η : N s m^{-2}

This expression for η only valid if the speed gradient v / l is uniform. If v / l is not uniform, then we must write

$$\eta = \frac{F / A}{dv / dy}$$

where $\frac{dv}{dy}$ is measured perpendicular to the direction of the velocity v.

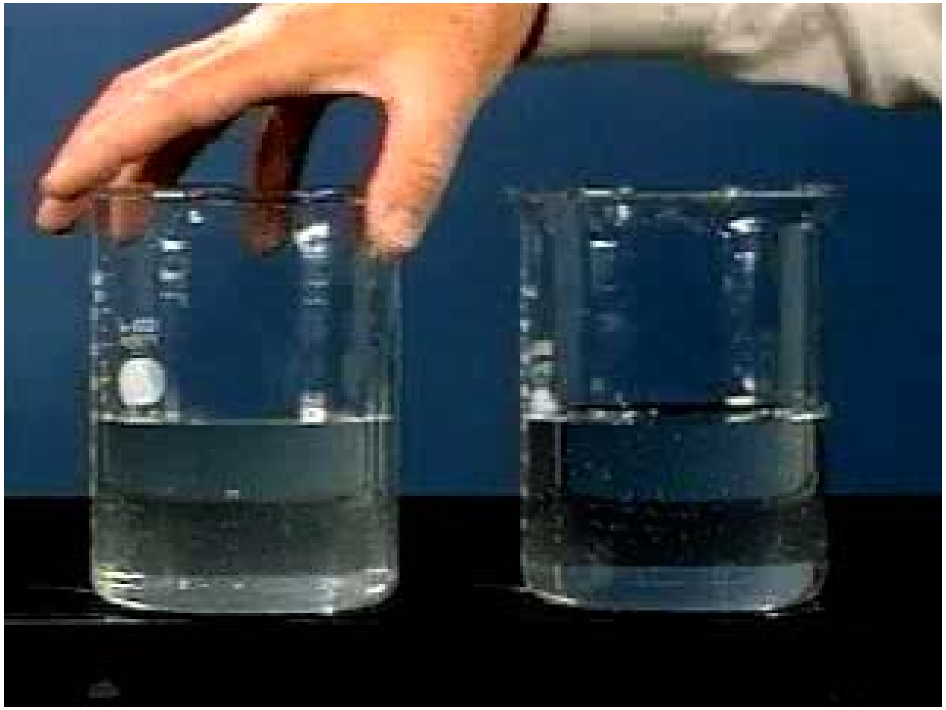
Viscosity ... 3

Fluids for which shearing stress is not linearly related to the rate of shearing strain are designated as non-Newtonian fluids.

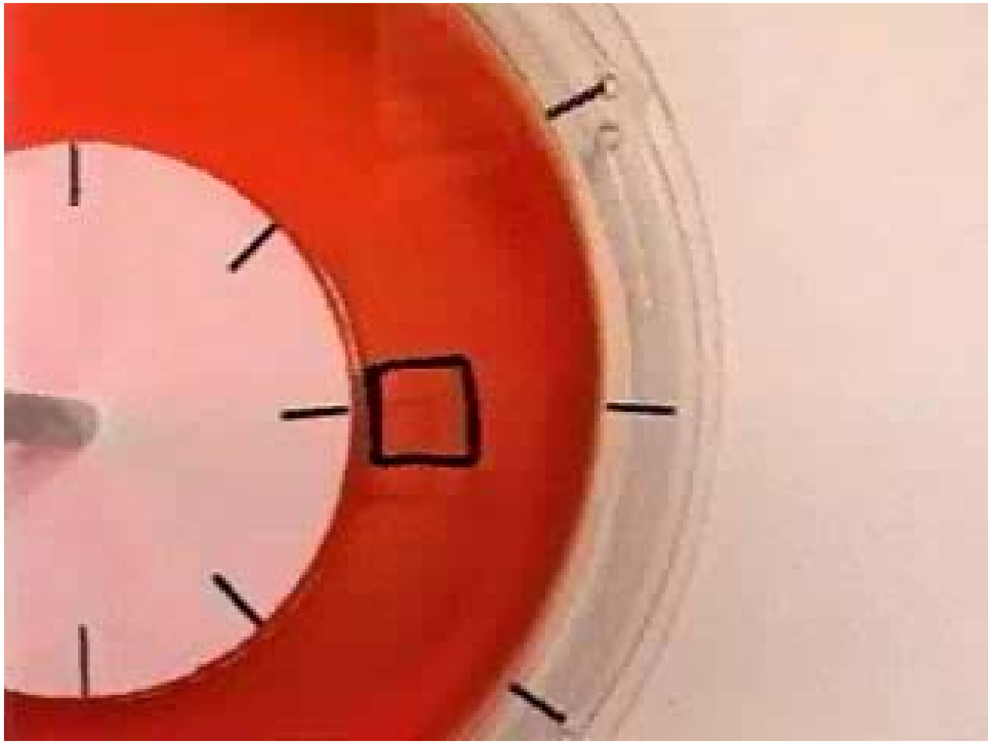
Here the viscous force has the form

$$\vec{\mathbf{f}}_{visc} = \eta \nabla^2 \vec{\mathbf{v}} + \left(\eta + \eta'\right) \vec{\nabla}\left(\vec{\nabla} \bullet \vec{\mathbf{v}}\right)$$

where η’ is the second coefficient of viscosity.



Video clip (1.1) Viscosity: Viscosity is responsible for the shear force produced in a moving fluid. Although these two fluids shown look alike (both are clear liquids and have a specific gravity of 1), they behave very differently when set into motion. The very viscous silicone oil is approximately 10,000 times more viscous than the water.



Video clip (6.1) Shear Deformation: Fluid elements located in a moving fluid move with the fluid and generally undergo a change in shape (angular deformation). A small rectangular fluid element is located in the space between concentric cylinders. The inner wall is fixed. As the outer wall moves, the fluid element undergoes an angular deformation. The rate at which the corner angles change (rate of angular deformation) is related to the shear stress causing the deformation.

The equations of motion for viscous (wet) fluids
The Navier-Stokes equation

In order to model the flow of real fluids, we need to include the phenomenon of viscosity.

Equation of motion:

$$\frac{\partial \vec{\mathbf{v}}}{\partial t} + \vec{\mathbf{v}} \bullet \vec{\nabla} \vec{\mathbf{v}} = -\frac{\vec{\nabla} p}{\rho} - \vec{\nabla}\phi + \frac{\vec{\mathbf{f}}_{visc}}{\rho}$$

where $\vec{\mathbf{f}}_{visc} = \underbrace{\eta \nabla^2 \vec{\mathbf{v}}}_{} + \underbrace{(\eta + \eta')\vec{\nabla}\left(\vec{\nabla} \bullet \vec{\mathbf{v}}\right)}_{}$

$$\eta\left(\frac{\partial^2 \vec{\mathbf{v}}}{\partial x^2} + \frac{\partial^2 \vec{\mathbf{v}}}{\partial y^2} + \frac{\partial^2 \vec{\mathbf{v}}}{\partial z^2}\right)$$

usually we ignore this term in the case of incompressible fluids

Therefore $\frac{D\vec{\mathbf{v}}}{Dt} = -\frac{\vec{\nabla} p}{\rho} - \vec{\nabla}\phi + \frac{\eta \nabla^2 \vec{\mathbf{v}}}{\rho}$

The Navier-Stokes equation ... 2

Therefore we have:

$$\frac{D\vec{\mathbf{v}}}{Dt} = -\frac{\vec{\nabla}p}{\rho} - \vec{\nabla}\phi + \frac{\eta\nabla^2\vec{\mathbf{v}}}{\rho}$$

where $\frac{D}{Dt} = \frac{\partial}{\partial t} + \vec{\mathbf{v}}\bullet\vec{\nabla}$ is the substantive derivative operator

This is the **Navier-Stokes equation.** You will find it appearing in a wide variety of contexts dealing with fluid flow.
When combined with the continuity equation of fluid flow $\left(\vec{\nabla}\bullet\vec{\mathbf{v}} = 0\right)$ it yields four equations in four unknowns: ρ and $\vec{\mathbf{v}}$

In most situations it is impossible to solve exactly, therefore approximations are usually made, or more recently, computer simulations are made.

The Navier-Stokes equation ... 3

Introducing the vorticity $\vec{\boldsymbol{\Omega}} = \vec{\nabla} \times \vec{\mathbf{v}}$ then

$$\rho\left\{\frac{\partial \vec{\mathbf{v}}}{\partial t} + \vec{\boldsymbol{\Omega}} \times \vec{\mathbf{v}} + \tfrac{1}{2}\vec{\nabla} v^2\right\} = -\vec{\nabla} p - \rho\, \vec{\nabla}\phi + \eta \nabla^2 \vec{\mathbf{v}}$$

Again assuming that the only body forces conservative forces such as gravity, and taking the curl on both sides, gives.

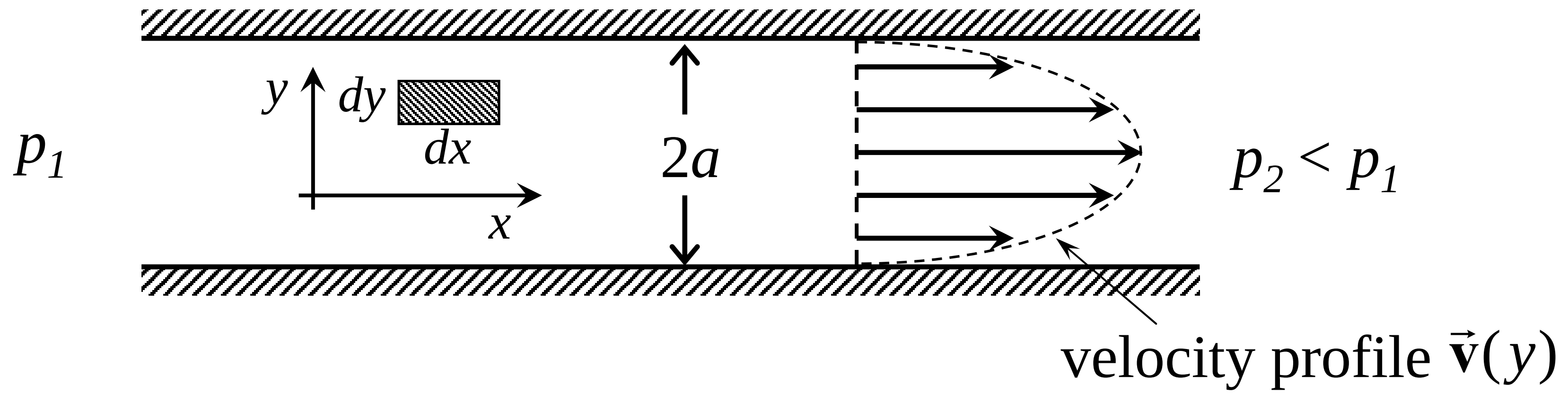
$$\frac{\partial \vec{\boldsymbol{\Omega}}}{\partial t} + \vec{\nabla} \times \left(\vec{\boldsymbol{\Omega}} \times \vec{\mathbf{v}}\right) = \frac{\eta\, \nabla^2 \vec{\boldsymbol{\Omega}}}{\rho}$$

This term was zero for the previous case of a "dry" fluid

If we disregard the term $\vec{\nabla} \times \left(\vec{\boldsymbol{\Omega}} \times \vec{\mathbf{v}}\right)$, the we have a diffusion equation.
The new term means that the vorticity diffuses through the fluid.
If there is a large gradient in the vorticity, then it will speed out into the neighbouring fluid.
The viscous equations can only be solved fully for a few special cases….

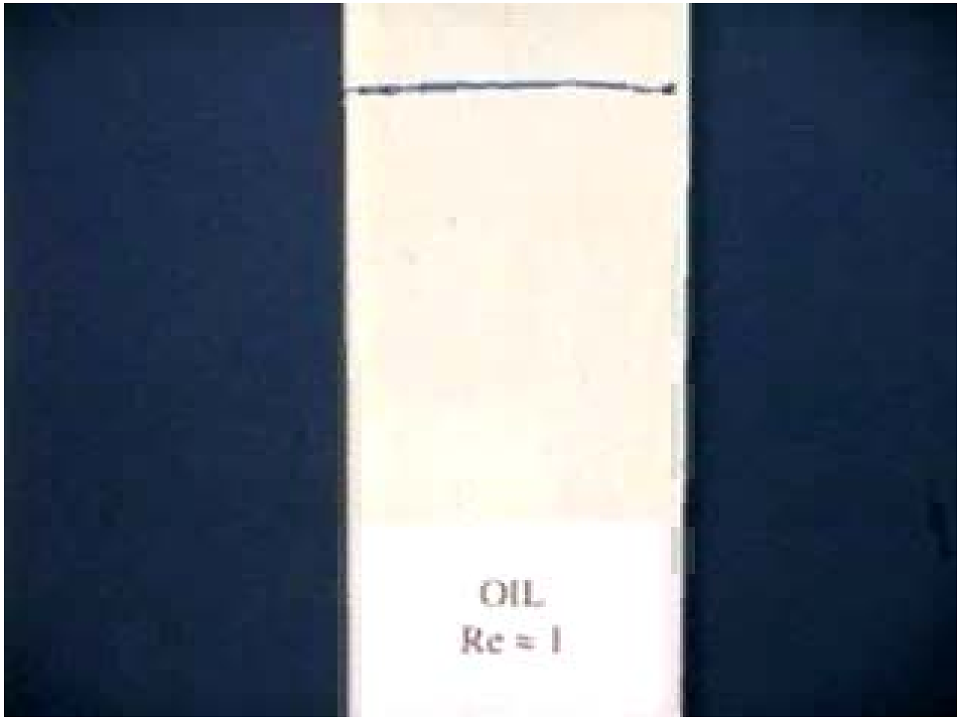
Examples of viscous, incompressible, laminar flow

A. channel flow (2D counterpart of pipe flow)

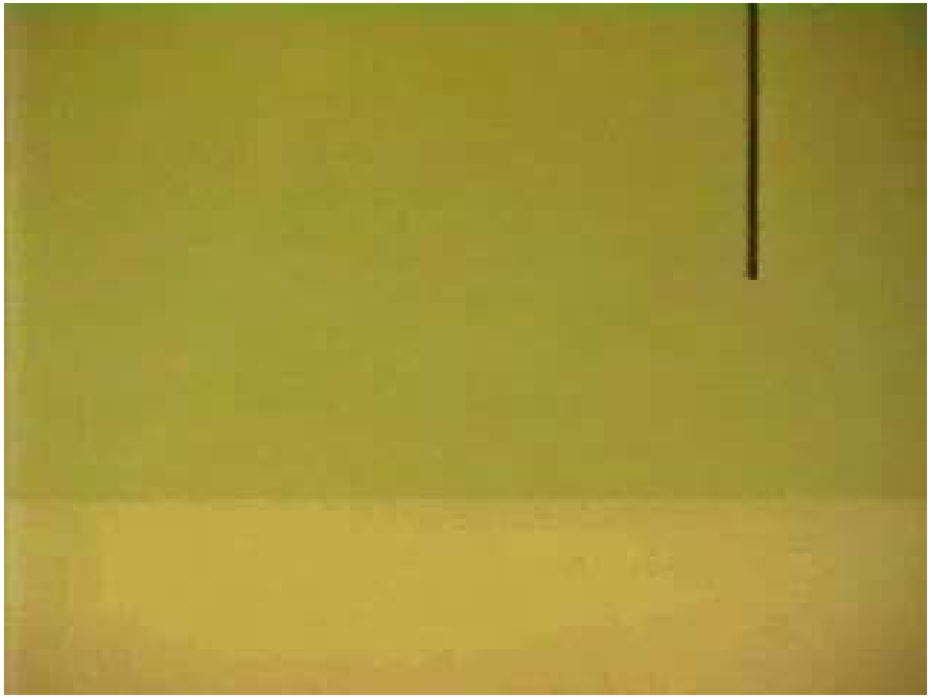


Conditions:

- $\vec{\mathbf{v}}(a) = \vec{\mathbf{v}}(-a) = 0$ ("no slip" condition)
- density ρ is constant everywhere
- velocity profile $\vec{\mathbf{v}}(y)$ is the same everywhere along the channel
- pressure p is a function of x, but not y.
- the coefficient of viscosity η is constant in space.



<u>Video clip (6.6) Laminar Flow</u>: The velocity distribution is parabolic for steady, laminar flow in circular tubes. A filament of dye is placed across a circular tube containing a very viscous liquid which is initially at rest. With the opening of a valve at the bottom of the tube the liquid starts to flow, and the parabolic velocity distribution is revealed. Although the flow is actually unsteady, it is quasi-steady since it is only slowly changing. Thus, at any instant in time the velocity distribution corresponds to the characteristic steady-flow parabolic distribution.



Video clip (1.2) No-Slip Condition: As a fluid flows near a solid surface, it "sticks" to the surface, i.e., the fluid matches the velocity of the surface. This so-called "no-slip" condition is a very important one that must be satisfied in any accurate analysis of fluid flow phenomena. Dye injected at the bottom of a channel through which water is flowing forms a stagnant layer near the bottom due to the no-slip condition. As the dye filament is moved away from the bottom, the motion of the water is clearly apparent. A significant velocity gradient is created near the bottom.

Video clip (6.5) No-Slip Boundary Condition: Boundary conditions are needed to solve the differential equations governing fluid motion. One condition is that any viscous fluid sticks to any solid surface that it touches. Clearly a very viscous fluid sticks to a solid surface as illustrated by pulling a knife out of a jar of honey. The honey can be removed from the jar because it sticks to the knife. This no-slip boundary condition is equally valid for small viscosity fluids. Water flowing past the same knife also sticks to it. This is shown by the fact that the dye on the knife surface remains there as the water flows past the knife.

Channel flow ... 2

Consider the forces acting on a small element of fluid of sides *dx* and *dy* (and *dz* in the *z*-direction)

Two sources of these forces: action of viscosity and pressure

Viscous force per unit cross sectional area = $\eta \dfrac{d\vec{\mathbf{v}}}{dy}$

Net viscous force on the fluid volume element

$$= \left[\eta \left(\frac{\partial \vec{\mathbf{v}}}{\partial y} \right)_{y+dy} - \eta \left(\frac{\partial \vec{\mathbf{v}}}{\partial y} \right)_{y} \right] dxdz$$

$$= \frac{\partial}{\partial y} \left(\eta \frac{\partial \vec{\mathbf{v}}}{\partial y} \right) dxdydz = \underbrace{\eta \frac{\partial^2 \vec{\mathbf{v}}}{\partial y^2}}_{\text{viscous force per unit volume}} dxdydz$$

Channel flow ... 3

Net pressure force acting on the volume element in the downstream direction = $(p)_x \, dydz \; - \; (p)_{x+dx} \, dydz$

$$(p)_x \, dydz \; - \left[(p)_x \; + \left(\frac{\partial p}{\partial x} \right) dx \right] dydz \quad = \; - \underbrace{\frac{\partial p}{\partial x}}_{\text{pressure force per unit volume}} dxdydz$$

The momentum of the element *dxdydz* is not changing
- each fluid particle travels downstream at a constant distance from the centre of the channel with a constant speed.

$$\sum \vec{\mathrm{F}}(\text{acting on the element}) = 0$$

$$\therefore \;\; \eta \frac{\partial^2 v}{\partial y^2} dxdydz \; - \; \frac{\partial p}{\partial x} dxdydz = 0$$

Can write $\quad -\dfrac{\partial p}{\partial x} = - \dfrac{dp}{dx} = \dfrac{p_1 - p_2}{l} \quad$ = a constant, G

since we have a constant velocity profile.

Channel flow ... 4

$$\therefore \quad \eta \frac{\partial^2 v}{\partial y^2} = -G$$

Integrate with the boundary condition
$v = 0$ at $y = \pm a$

$$v(y) = \frac{G}{2\eta}\left(a^2 - y^2\right)$$ ... a parabola

with $v_{\max} = G\dfrac{a^2}{2\eta}$

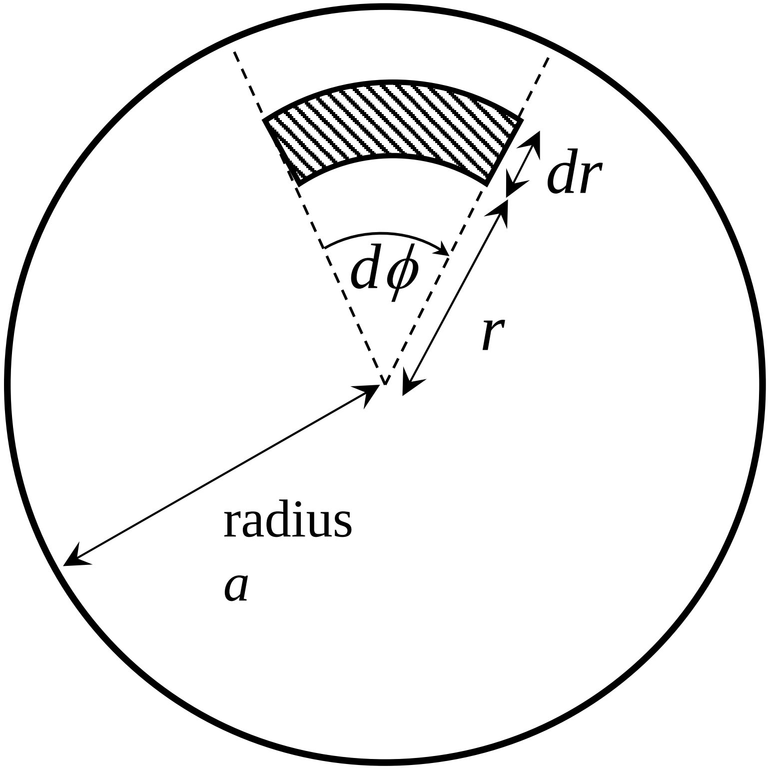
The mass of a fluid passing through the channel per unit time per unit length in the *z*-direction = $\int_{a}^{-a} \rho v dy = \ \ldots \ = \dfrac{2G\rho a^3}{3\eta}$

<u>Video clip (10.3) Uniform Channel Flow:</u> The Manning Equation is used to determine the constant-depth flowrate in a straight channel with constant slope and constant cross-section. Although the geometry for many man-made channels is sufficiently uniform to allow the use of the Manning equation, the irregularity of many channels (especially natural ones) makes the use of the Manning equation a rough approximation at best. Curves in the channel, variable flowrate along the channel (as with rainwater runoff into a gutter), or irregular channel cross-section may cause the calculated flowrates to be quite different than the actual flowrate.

B. Laminar flow in a pipe

(also known as Poiseuille flow)

Cross section of pipe:



Use cylindrical coordinate system

Flow direction (z-axis) is normal to the page.

$\vec{\mathbf{v}} = \vec{\mathbf{v}}(r)$ (independent of z)

Conditions:

- $\vec{\mathbf{v}}(a) = 0$ ("no slip" condition)
- density ρ is constant everywhere
- velocity profile $\vec{\mathbf{v}}(r)$ is the same everywhere along the pipe
- pressure p is a function of z *only*.
- the coefficient of viscosity η is constant in space.

Laminar flow in a pipe ... 2

Net viscous force on fluid volume element $rd\phi drdz$:

$$= \left[\eta \left(\frac{\partial v}{\partial r} \right)_{r+dr} - \eta \left(\frac{\partial v}{\partial r} \right)_{r} \right] rd\phi dz$$

$$= \left[\eta \left(\frac{\partial v}{\partial r} \right)_{r} + \eta \frac{\partial}{\partial r} \left(r \frac{\partial v}{\partial r} \right)_{r} \frac{dr}{r} - \eta \left(\frac{\partial v}{\partial r} \right)_{r} \right] rd\phi dz$$

$$= \eta \frac{\partial}{\partial r} \left(r \frac{\partial v}{\partial r} \right)_{r} drd\phi dz$$

Net pressure force on fluid volume element $rd\phi drdz$:

$$= \left(p \right)_{x} rd\phi dr - \left(p \right)_{x+dx} rd\phi dr$$

$$= \left(p \right)_{x} rd\phi dr - \left[\left(p \right)_{x} + \left(\frac{\partial p}{\partial z} \right) dz \right] rd\phi drdz$$

$$= - \left(\frac{\partial p}{\partial z} \right) rdrd\phi dz$$

Laminar flow in a pipe ... 3

Again $\sum \vec{\mathbf{F}} = 0$ since $\frac{d}{dt}(\text{momentum}) = 0$

$$\therefore \eta \frac{\partial}{\partial r}\left(r\frac{\partial v}{\partial r}\right)_r dr d\phi dz - \left(\frac{\partial p}{\partial z}\right) r dr d\phi dz = 0$$

$$-\frac{\partial p}{\partial z} = -\frac{dp}{dz} = G \text{ (a constant)}$$

$$\therefore \eta \frac{d}{dr}\left(r\frac{dv}{dr}\right) = -rG$$

integrate

$$\therefore v = -\frac{Gr^2}{4\eta} + A\ln r + B$$

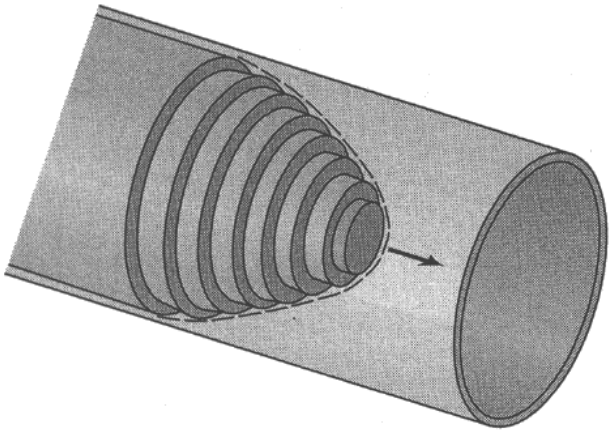
$$B = \frac{Ga^2}{4\eta}$$

since $v = 0$ at $r = a$

A must be zero, since *r* cannot be infinite

Laminar flow in a pipe ... 4

$$\therefore \; v(r) = -\frac{G}{4\eta}\left(a^2 - r^2\right)$$ ... a paraboloid



with $$v_{\max} = \frac{Ga^2}{4\eta}$$

The mass per unit time (mass flux) passing through the pipe is

$$\int_0^a \rho 2\pi r v \, dr = \frac{\pi\rho G a^4}{8\eta} = \frac{\pi\rho\left(p_1 - p_2\right)a^4}{8\eta l}$$

where $p_1 - p_2$ is the pressure difference across a length l of the pipe .

Video clip (8.1) Laminar/Turbulent Pipe Flow: Whether flow in a pipe is laminar, transitional, or turbulent depends on the value of the Reynolds number. In this experiment water flows through a clear pipe with increasing speed. Dye is injected through a small diameter tube at the left portion of the screen. Initially, at low speed (Re <2100) the flow is laminar and the dye stream is stationary. As the speed (Re) increases, the transitional regime occurs and the dye stream becomes wavy (unsteady, oscillatory laminar flow). At still higher speeds (Re>4000) the flow becomes turbulent and the dye stream is dispersed randomly throughout the flow.

Viscous flow past a circular cylinder

Consider the flow of an incompressible, viscous fluid past a long cylinder of diameter *D*.

The flow is described by

- $$\frac{\partial \vec{\Omega}}{\partial t} + \vec{\nabla} \times \left(\vec{\Omega} \times \vec{\mathbf{v}} \right) = \frac{\eta \nabla^2 \vec{\Omega}}{\rho}$$
- $\vec{\Omega} = \vec{\nabla} \times \vec{\mathbf{v}}$
- $\vec{\nabla} \bullet \vec{\mathbf{v}} = 0$

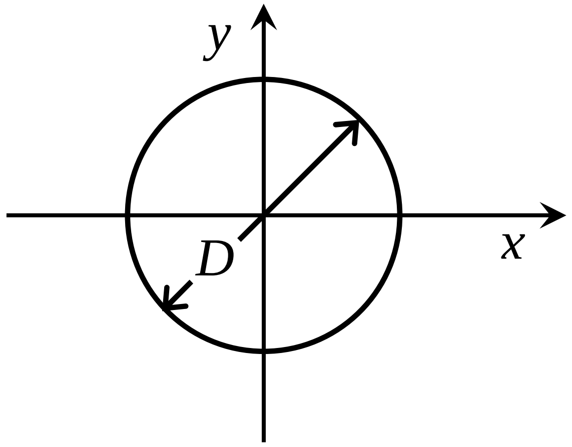
with the conditions:

• the velocity at large distances is constant, say *V* and parallel to the *x*-axis.

• the velocity at the surface of the cylinder is zero.

i.e. $v_x = v_y = v_z = 0$

for $x^2 + y^2 = \dfrac{D^2}{4}$



Viscous flow past a circular cylinder ... 2

There are 4 different parameters: η, ρ, D and V.

However, we can show that all the possible solutions correspond to different values of **one** parameter

Firstly the ratio $\dfrac{\eta}{\rho}$ (= the specific viscosity) appears only **once**.

Secondly we measure all distance in terms of the only length that appears - the diameter D.

We then write $x = x'D$, $y = y'D$, $z = z'D$.

In the same way, we measure all velocities in terms of V, by setting $v = v'V$ where $v' = 1$ at large distances.

Our unit of time is then $\dfrac{D}{V}$ and we set $t = t'\dfrac{D}{V}$

Viscous flow past a circular cylinder ... 3

The derivatives also change from $\frac{\partial}{\partial x} \rightarrow \frac{1}{D}\frac{\partial}{\partial x'}$ etc.

and our main equation is written:

$$\frac{\partial \vec{\Omega}'}{\partial t} + \vec{\nabla}' \times \left(\vec{\Omega}' \times \vec{\mathbf{v}}'\right) = \frac{\eta}{\rho V D} \nabla^2 \vec{\Omega}'$$

We introduce the **Reynolds number** $\mathrm{Re} = \frac{\rho}{\eta} V D$

If we remember that all our equations are to be written with all quantities in the new units, the primes may be omitted, giving:

$$\frac{\partial \vec{\Omega}}{\partial t} + \vec{\nabla} \times \left(\vec{\Omega} \times \vec{\mathbf{v}}\right) = \frac{1}{\mathrm{Re}} \nabla^2 \vec{\Omega} \quad \text{and} \quad \vec{\Omega} = \vec{\nabla} \times \vec{\mathbf{v}}$$

with the conditions:

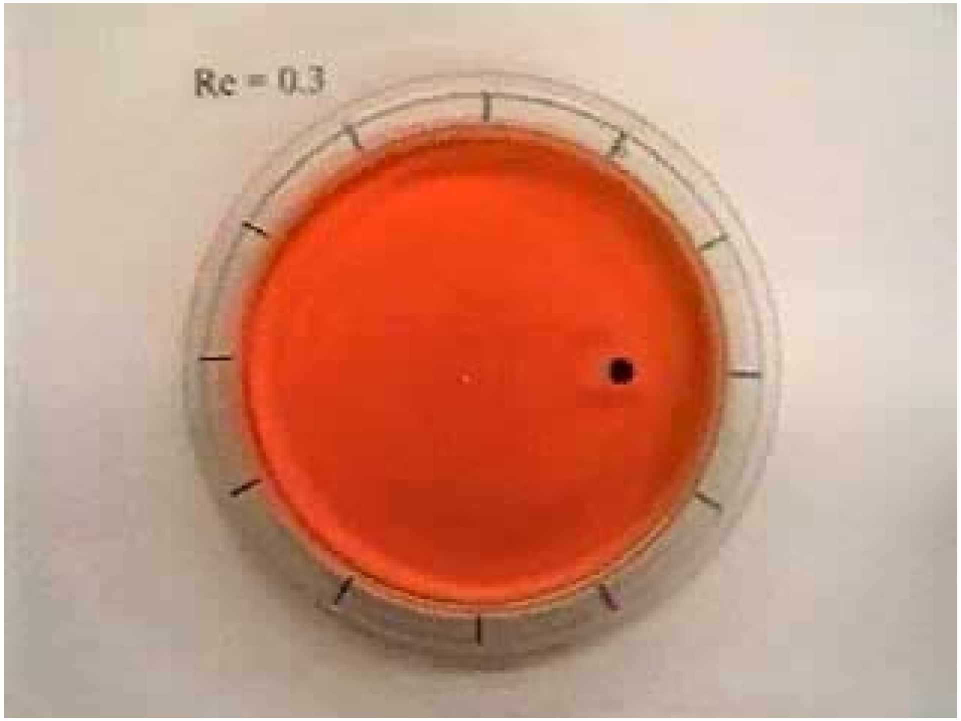
- $\vec{\mathbf{v}} = 0$ for $x^2 + y^2 = \frac{1}{4}$
- $v_x = 1\,,\ v_y = v_z = 0$ for $x^2 + y^2 + z^2 >> 1$

The Reynolds Number

The Reynolds number is undoubtedly the most famous dimensionless parameter in fluid mechanics which can be used as a criterion to distinguish between **laminar** and **turbulent** flow. In most fluid flow problems there will be a characteristic length, D , and a velocity, V, as well as the fluid properties of density, ρ , and viscosity, η , which are relevant variables in the problem. Thus, with these variables the Reynolds number

$$\mathrm{Re} = \frac{\rho}{\eta} VD$$

arises naturally from the dimensional analysis. The Reynolds number is a measure of the ratio of the inertia force on an element of fluid to the viscous force on an element. When these two types of forces are important in a given problem, the Reynolds number will play an important role. However, if the Reynolds number is very small (Re ~ 1), this is an indication that the viscous forces are dominant in the problem, and it may be possible to neglect the inertial effects; that is, the density of the fluid will not be an important variable. Flows at very small Reynolds numbers are commonly referred to as “creeping flows”. Conversely, for large Reynolds number flows, viscous effects are small relative to inertial effects and for these cases it may be possible to neglect the effect of viscosity and consider the problem as one involving a “non-viscous” fluid.



<u>**Video clip (7.1) Reynolds Number:**</u> Important dimensionless groups can frequently be given useful physical interpretations. For example, the Reynolds number is an index of the ratio of inertia forces to viscous forces in a moving fluid. For a rotating tank containing a very viscous fluid, which gives a small Reynolds number, viscous forces are dominant. Thus, when the tank is suddenly stopped fluid particles also suddenly stop due to the dominance of viscous forces over inertia forces. Correspondingly, when a low viscosity fluid is in the tank, which gives a much higher Reynolds number, inertia forces are dominant. When the tank suddenly stops the fluid particles continue to move.

Viscous flow past a circular cylinder ... 4

Our equations for flow around the cylinder are very interesting. It means, for example, that if we solve the problem of the flow for one velocity V_1 and certain diameter D_1, and then ask about the flow for a different diameter D_2 and a different fluid, the flow will be the same for the velocity V_2 which gives the **same** Reynolds number

i.e. when $$\mathrm{Re}_1 = \frac{\rho_1}{\eta_1} V_1 D_1 = \mathrm{Re}_2 = \frac{\rho_2}{\eta_2} V_2 D_2$$

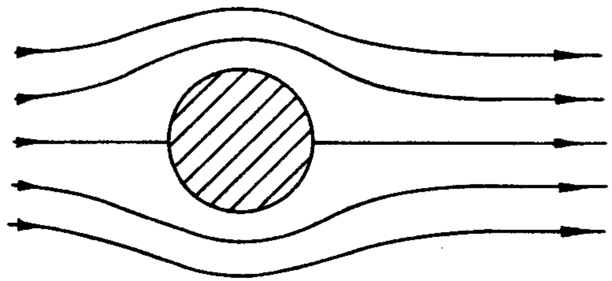
For any two situations which have the same Reynolds number, the flows will "look" the same in terms of the appropriately scaled x', y', z', and t'.
This principle allows the application of "wind tunnel" measurements on small scale airplanes, or "model basin" results on scale model boats, to the full scale objects.
(... provided that the compressibility of the fluid can be neglected.)

Viscous flow past a circular cylinder ... 5

The flow patterns of a fluid around a cylinder is one of the simplest situations, but only for low Reynolds numbers can the flow as a whole be determined analytically, due to the onset of **turbulence**.

The following description of the flow patterns is based almost entirely on experimental observations. Fig. (a) shows the flow when Re ~ 1. The lines indicate the paths of elements of fluid. The flow shows no unexpected properties, but two points are worth noting for comparison with higher values of the Reynolds number. Firstly, the flow is symmetrical upstream and downstream; the right-hand half of Fig. (a) is the mirror image of the left-hand half. Secondly, the presence of the cylinder has an effect over large distances; even many diameters to one side the velocity is appreciably different from V.

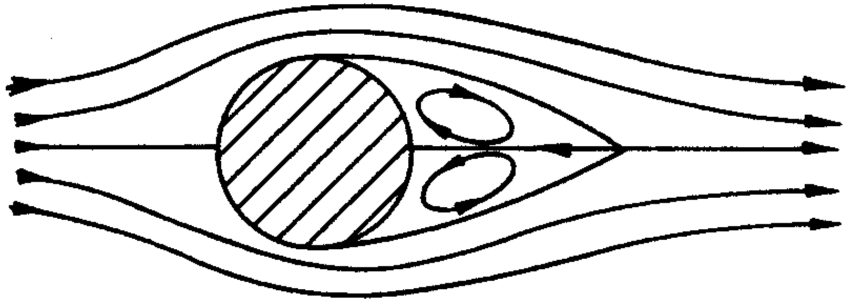
(a)



Viscous flow past a circular cylinder ... 6

As Re is increased the upstream-downstream symmetry disappears. The particle paths are displaced by the cylinder for a larger distance behind it than in front of it. When Re exceeds about 4, this leads to the feature shown in the flow pattern of Fig. (b). There is a circulation behind the sphere. Fluid that comes round the cylinder close to it moves away from it before reaching the rear point of symmetry. As a result, two "attached eddies" exist behind the cylinder; the fluid in these circulates continuously, not moving off downstream. These eddies get bigger with increasing Re. It is still an open question as to whether there is always a circulation there even at the smallest Reynolds number or whether things suddenly change at a certain Reynolds number. It used to be thought that the circulation grew continuously. But it is now thought that it appears suddenly, and it is certain that the circulation increases with Re. In any case, there is a different character to the flow for Re in the region from about 10 to 30. There is a pair of vortices behind the cylinder.

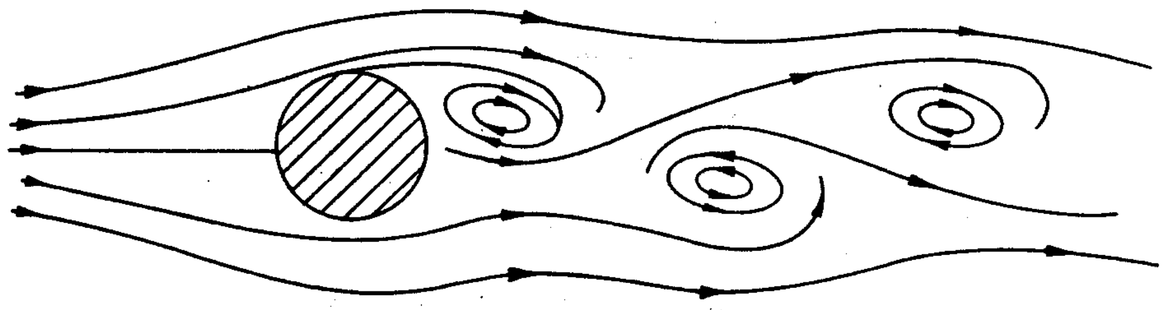
(b)



Viscous flow past a circular cylinder ... 7

The tendency for the most striking flow features to occur downstream of the cylinder becomes even more marked as one goes to higher Reynolds numbers. This region is called the wake of the cylinder. For Re > 40, the flow becomes unsteady and there is suddenly a complete change in the character of the motion. As with transition to turbulence in a pipe, this unsteadiness arises spontaneously even though all the imposed conditions are being held steady. The instability develops to give the flow pattern, known as a Kármán vortex street, shown schematically in Fig. (c). What happens is that one of the vortices behind the cylinder gets so long that it breaks off and travels downstream with the fluid. Then the fluid curls around behind the cylinder and makes a new vortex. The vortices peel off alternately on each side, so an instantaneous view of the flow looks roughly as sketched in Fig. (c).

(c)



Viscous flow past a circular cylinder ... 8

Concentrated regions of rapidly rotating fluid (regions of locally high vorticity) form two rows on either side of the wake. All the vortices on one side rotate in the same sense, those on opposite sides in opposite senses. Longitudinally, the vortices on one side are mid-way between those on the other.

When Re is greater than about 100, the attached eddies are periodically shed from the cylinder to form the vortices of the street. Whilst the eddy on one side is being shed that on the other side is re-forming. The difference between the two flows in Fig (c) and Fig. (b) is almost a complete difference in regime. In Fig. (a) or (b), the velocity is constant, whereas in Fig (c), the velocity at any point varies with time. There is no steady solution above Re = 40. For these higher Reynolds numbers, the flow varies with time but in a regular, cyclic fashion. We can get a physical idea of how these vortices are produced. We know that the fluid velocity must be zero at the surface of the cylinder and that it also increases rapidly away from that surface. Vorticity is created by this large local variation in fluid velocity. Now when the main stream velocity is low enough, there is sufficient time for this vorticity to diffuse out of the thin region near the solid surface where it is produced and to grow into a large region of vorticity.

Viscous flow past a circular cylinder ... 9

The exact behaviour of a Kármán vortex street is very sensitive to disturbances and so it is difficult to predict just what will occur at any given Reynolds number. One can, however, say that the lower the Reynolds number the greater is the likelihood of straight, parallel vortices.

With increasing Re the strong regularity of the velocity variation is lost. An important transition occurs at a Reynolds number a little below 200. Below this, the vortex street continues to all distances downstream. Above, it breaks down and produces ultimately a turbulent wake, although the intermediate stages take different forms over different Reynolds number ranges. The name turbulent implies the existence of highly irregular rapid velocity fluctuations. The turbulence is confined to the long narrow wake region downstream of the cylinder. The change at Re ~ 200 arises from instability of the vortex street to three-dimensional disturbances. Irregularities in the vortices increase in amplitude as the vortices travel downstream. Ultimately the irregularities become dominant and the wake is turbulent.

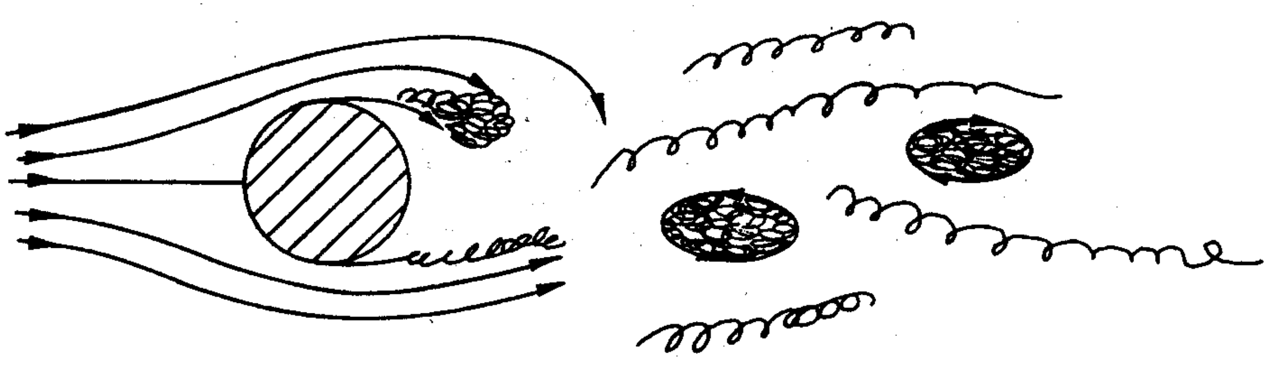
Viscous flow past a circular cylinder ... 10

At Re ~ 400, a further instability occurs. As the vortices travel downstream the turbulence spreads into the regions between them and disrupts the regular periodicity, leading finally to a fully turbulent wake. This behaviour continues over a wide Reynolds number range, up to about 3×10^5. At Re ~ 3×10^5, a much more dramatic development occurs. To understand this we must first consider developments at lower Re at the front and sides of the cylinder. There the phenomenon known as boundary layer formation occurs. There is a region, called the boundary layer, next to the wall of the cylinder in which all the changes to the detailed flow pattern occur. Outside this the flow pattern is independent of the Reynolds number. For these statements to mean anything, the boundary layer must be thin compared with the diameter of the cylinder; this is the case when Re is greater than about 100.

Viscous flow past a circular cylinder ... 11

The change in the flow at Re ~ 3×10^5 results from developments in the boundary layer. Below this Reynolds number the motion there is laminar. Above it undergoes transition to turbulence as shown in Fig. (d). At first, this transition takes a rather complicated form. Laminar fluid close to the wall moves away from it as if it were entering the attached eddies; transition then occurs very quickly and the turbulent flow reattaches to the wall only a small distance downstream from the laminar separation. There are also complications due to the facts that the transition can occur asymmetrically between the two sides of the cylinder and non-uniformly along its length. In the turbulent region, the velocities are very irregular and "noisy". The flow is also no longer two-dimensional but twists and turns in all three dimensions. There is still a regular alternating motion superimposed on the turbulent one.

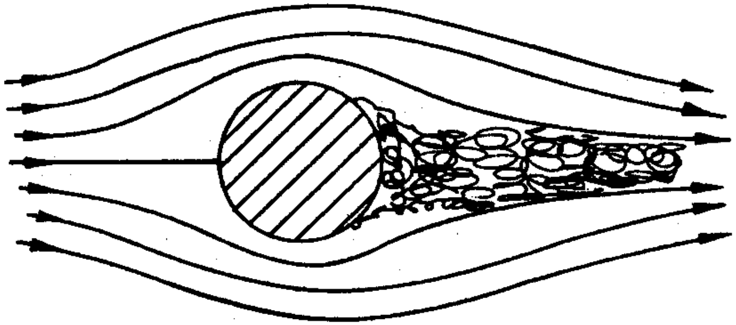
(d)



Viscous flow past a circular cylinder ... 12

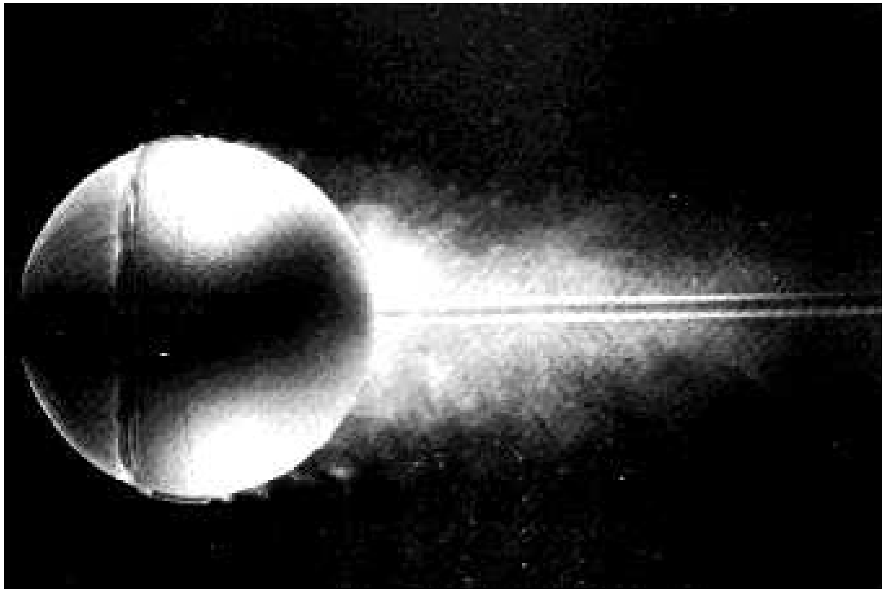
At higher values of the Reynolds number, above about 3×10^6, transition occurs in the boundary layer itself, thus eliminating the laminar separation and turbulent reattachment (see Fig (e)). The transition process is now similar to that described for pipe flow. Whether or not it has previously undergone laminar separation and turbulent reattachment, the turbulent boundary layer itself separates; the fluid in it moves away from the wall of the cylinder and into the wake some distance before the rear line of symmetry. This occurs, however, much further round the cylinder than when the boundary layer remains laminar. As a result the wake is narrower for $Re > 3 \times 10^5$ than for $Re < 3 \times 10^5$. When $Re > 3 \times 10^5$, the fluid entering the wake is already turbulent and so the transition process immediately behind the cylinder is eliminated.

(e)

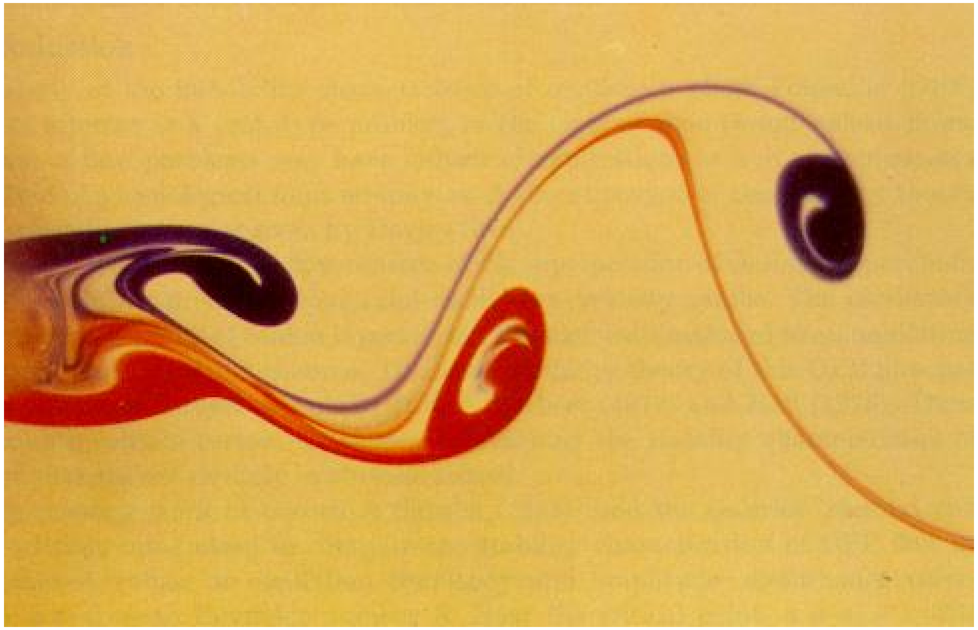


Viscous flow past a circular cylinder ... 13

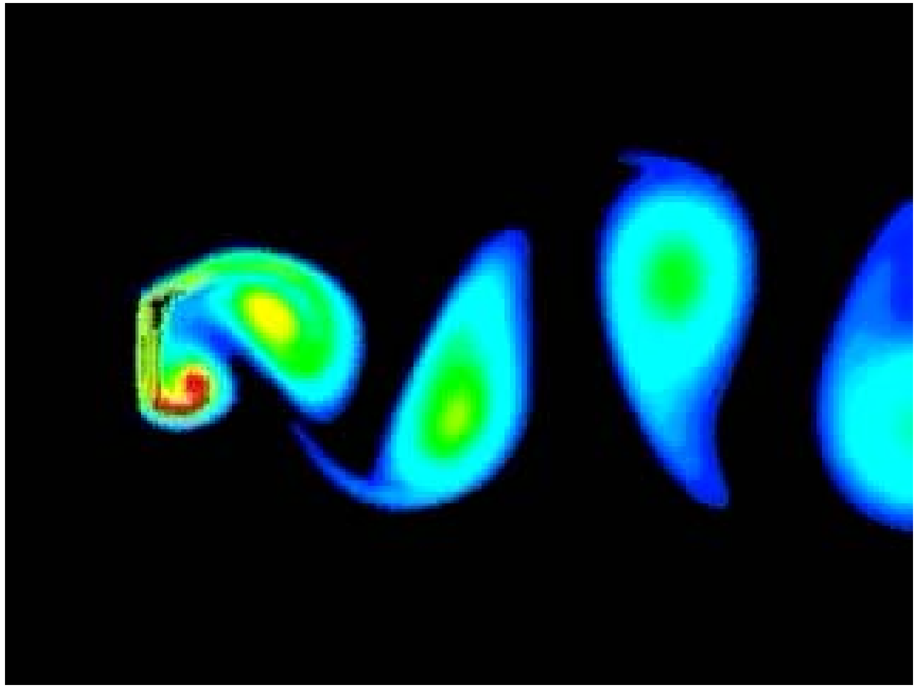
What happens for still larger Reynolds numbers? As we increase the speed further, the wake increases in size again and the drag increases. The latest experiments, which go up to Re = 10^7 or so, indicate that a new periodicity appears in the wake, either because the whole wake is oscillating back and forth in a gross motion or because some new kind of vortex is occurring together with an irregular noisy motion.



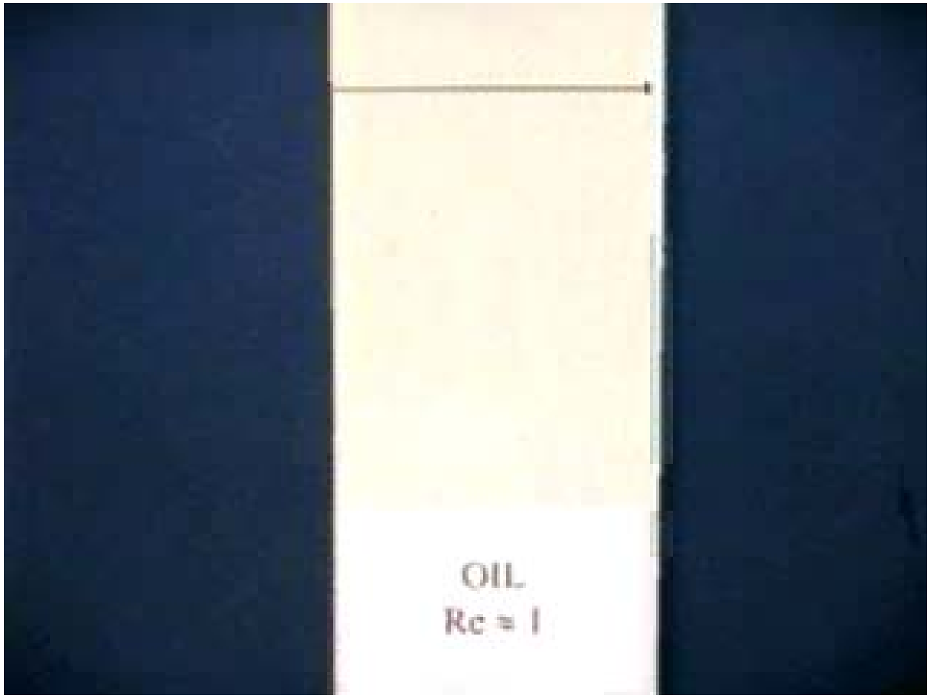
29. A Round Jet in Cross-Flow



Kármán vortex street

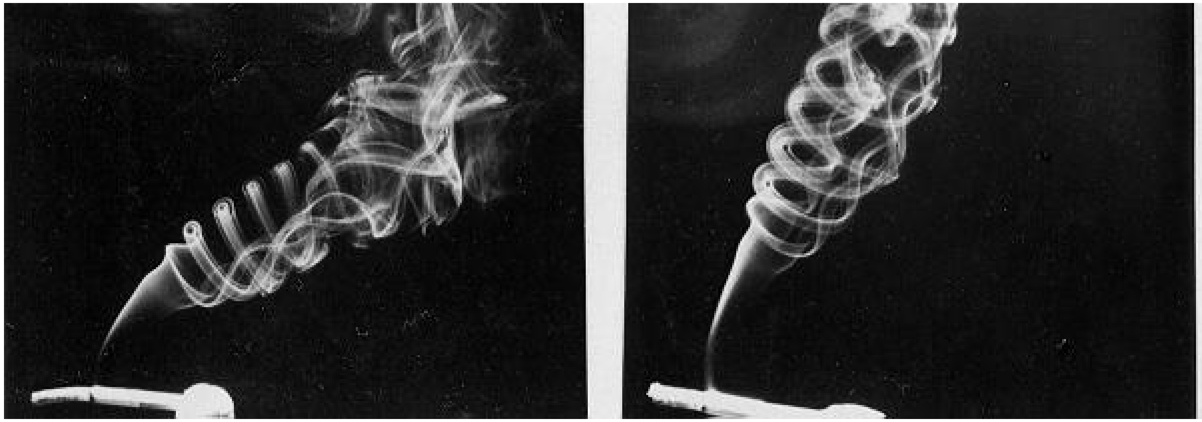


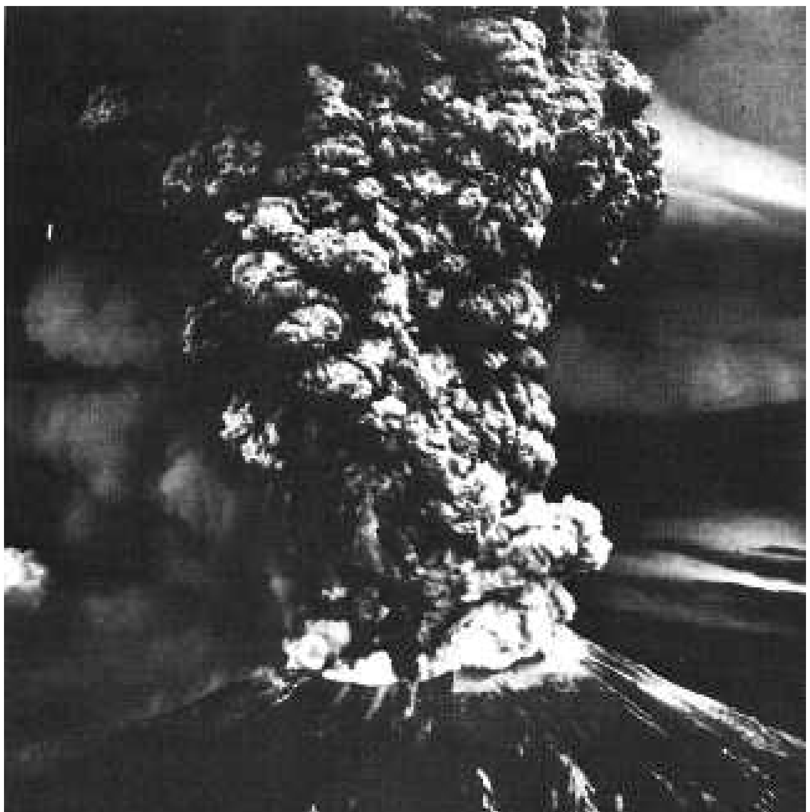
<u>Video clip (9.6) Oscillating Sign:</u> Steady flow past a blunt object may produce an oscillating Karman vortex street wake behind the object. The shedding of vortices from the body exerts a periodic force on the body that, if the frequency is right, can cause the body to oscillate. As shown in a computational fluid dynamics simulation, flow past a flat plate can produce a well-defined Karman vortex street. Similarly, wind blowing past a rectangular speed limit sign can excite the sign into significant twisting motion, provided the conditions (wind speed and direction, stiffness of the sign support, etc.) are correct.



Video clip (8.3) Laminar/Turbulent Velocity Profiles: The velocity profile for laminar flow in a pipe is quite different than that for turbulent flow. An approximation to the velocity profile in a pipe is obtained by observing the motion of a dye streak placed across the pipe. With a viscous oil at Reynolds number of about 1, viscous effects dominate and it is easy to inject a relatively straight dye streak. The resulting laminar flow profile is parabolic.

With water at Reynolds number of about 10,000, inertial effects dominate and it is difficult to inject a straight dye streak. It is clear, however, that the turbulent velocity profile is not parabolic, but is more nearly uniform than for laminar flow.

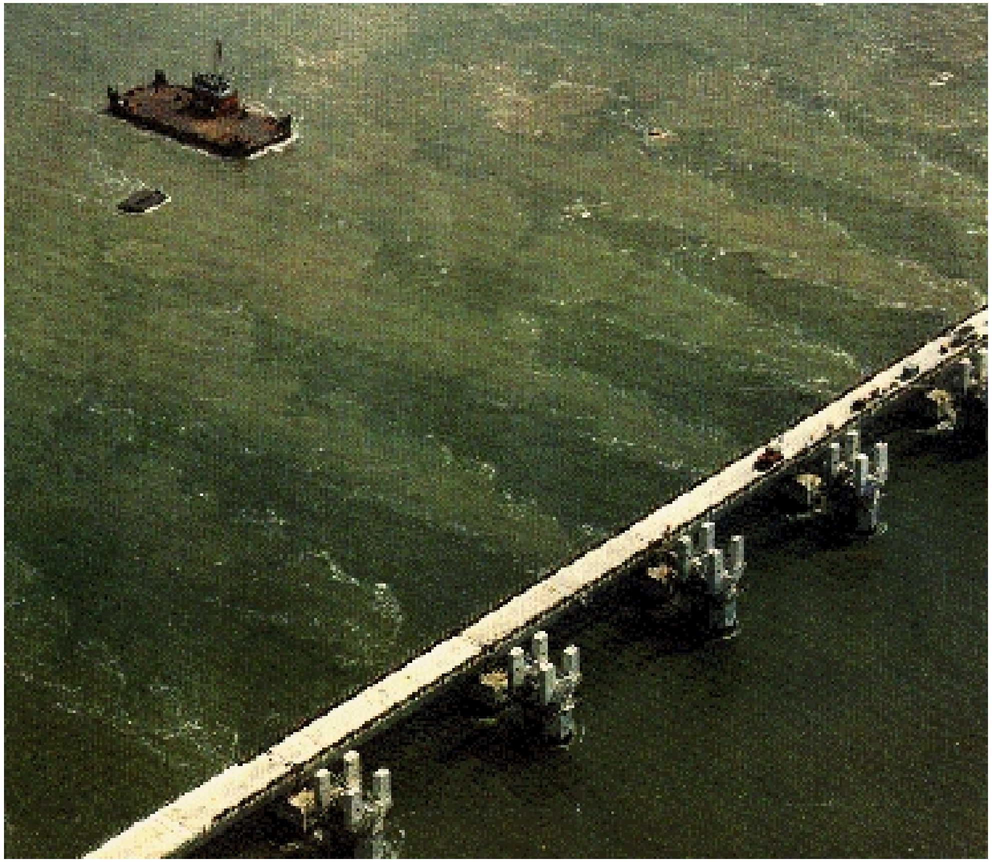


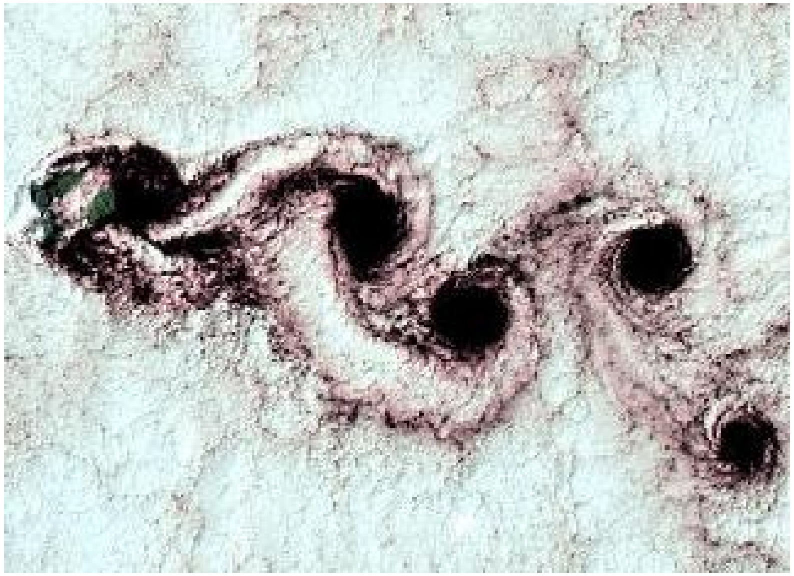


Turbulence



Trailing vortices

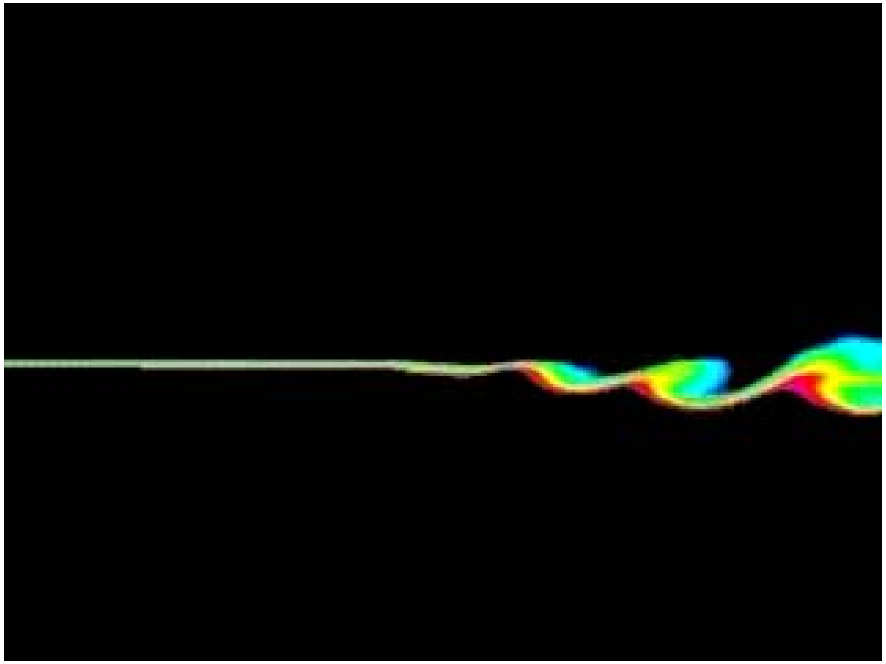




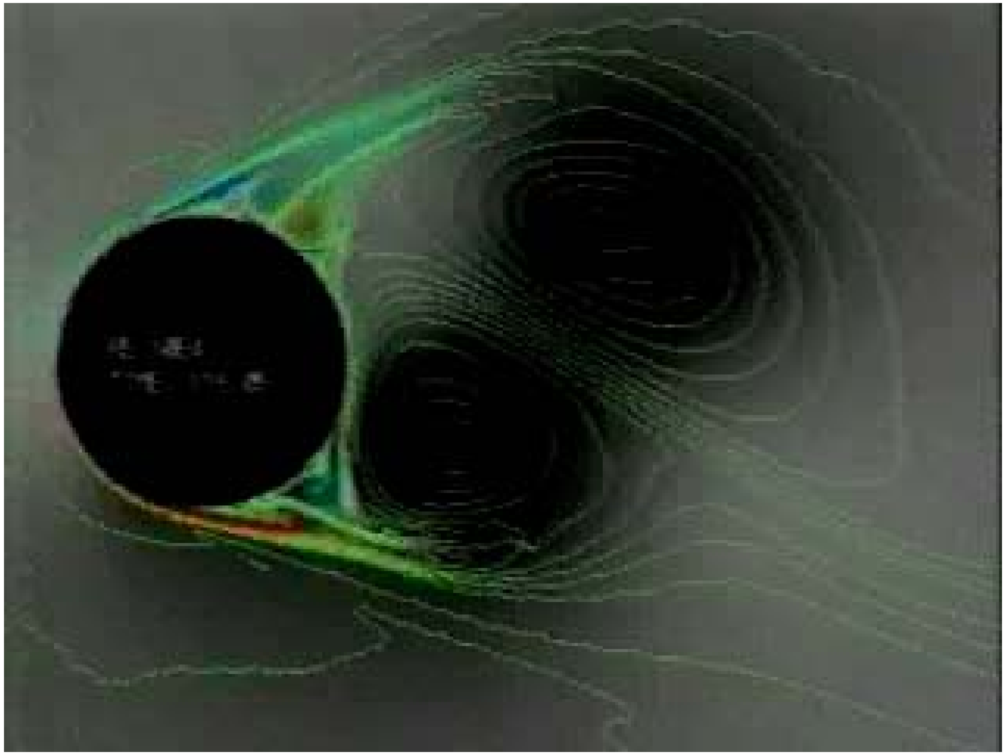
Trailing vortices


K-H Instability Cloud
PSC Cloud Photo
Courtesy of James D. Rufo

Video clip (4.2) Flow Past a Wing: Most flows involve complex, three-dimensional, unsteady conditions. The flow past an airplane wing provides an example of these phenomena. The flow generated by an airplane is made visible by flying a model Airbus airplane through two plumes of smoke. The complex, unsteady, three-dimensional swirling motion generated at the wing tips (called trailing vorticies) is clearly visible. An understanding of this motion is needed to ensure safe flying conditions, especially during landing and take-off operations where it can be dangerous for an airplane to fly into the preceding airplane's trailing vorticies.



Video clip (9.3) Laminar/Turbulent Transition: Near the leading edge of a flat plate, the boundary layer flow is laminar. If the plate is long enough, the flow becomes turbulent, with random, irregular mixing. A similar phenomenon occurs at the interface of two fluids moving with different speeds. As shown in a computational fluid dynamic simulation, the interface between two fluids moving horizontally with different speeds becomes unstable and waves develop on the surface. Similarly, the rising smoke plume from a cigarette is laminar near the source, becomes wavy at a certain location, and then breaks into turbulence.

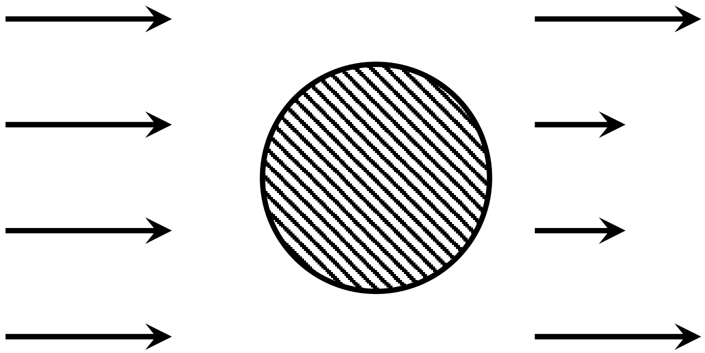


Video clip (6.7) CFD example: Complex flows can be analyzed using the finite difference method in which the continuous variables are approximated by discrete values calculated at grid points. The governing partial differential equations are reduced to a set of algebraic equations which is solved by approximate numerical methods. The flow past a circular cylinder at a Reynolds number of 10,000 was calculated using a finite difference method based on the Navier-Stokes equations. The O-shaped grid contained 100 by 400 grid points. Particle paths clearly show the Karmen vortex street.

Drag

Drag is an important quantity associated with the relative motion between a body and a fluid is the force produced on the body. One has to apply a force in order to move a body at constant speed through a stationary fluid. Correspondingly an obstacle placed in a moving fluid would be carried away with the flow if no force were applied to hold it in place. The force in the flow direction exerted by the fluid on an obstacle is known as the **drag**. Because of the force between it and the obstacle, the momentum is removed from the fluid.

The rate of momentum transport downstream must be smaller behind the obstacle than in front of it. There is a reduction in the velocity in the wake region.



Drag ... 2

When the velocity is very low (e.g. in cases of high viscosity) the inertial forces are negligible and the flow is described by the equation:

$$\nabla^2 \vec{\Omega} = 0$$

This equation was first solved by Stokes.

For a small sphere moving under conditions of low Re, the drag force

$$F = 6\pi\eta a V$$

where a = radius of the sphere and V is the speed of the sphere

This is very useful for centifuges, sedimentation and diffusion.

Drag ... 2

Introduce the drag coefficient C_D:

$$C_D = \frac{F}{\frac{1}{2}\rho V^2 D l}$$

where:

F: drag force ,

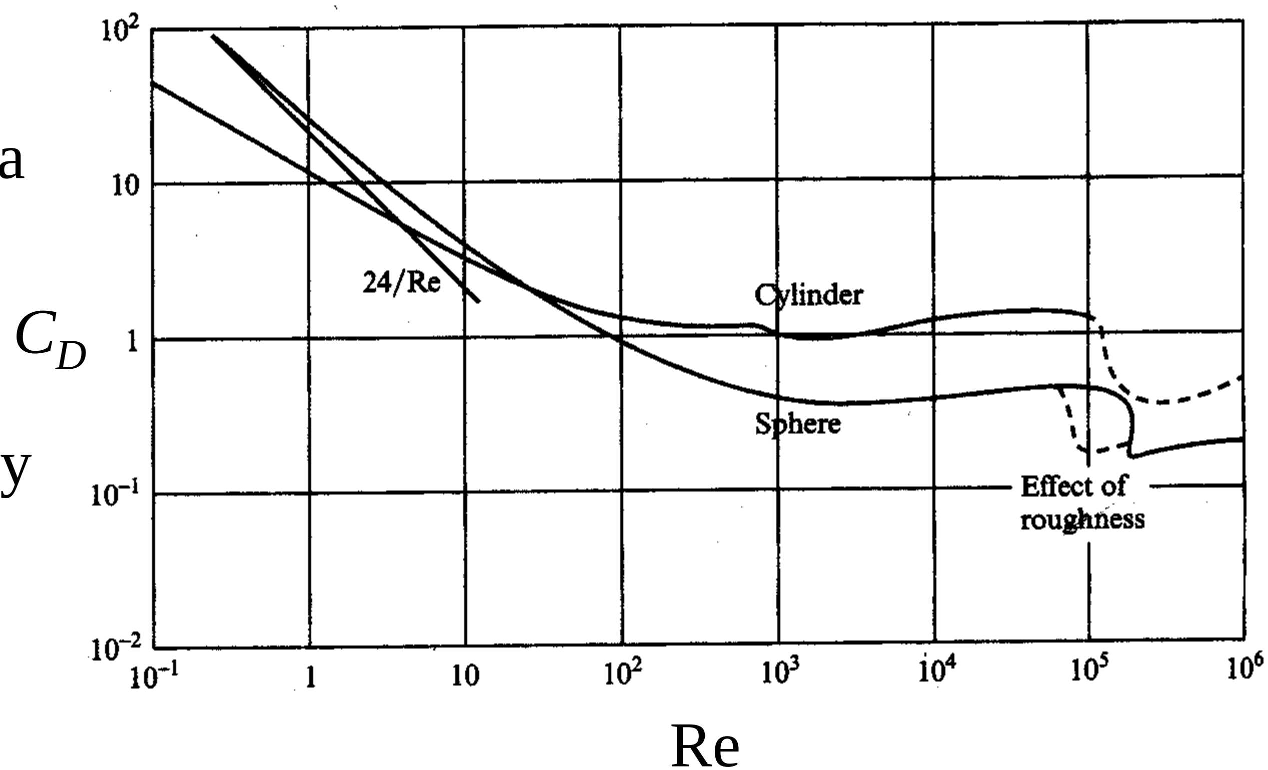
D : diameter of the cylinder ,

l : length of the cylinder ,

ρ : density of the fluid

Curve shows the variation of C_D as a function of Re.

- based on experimental measurements, only at low Re can the experiments be matched to theory.

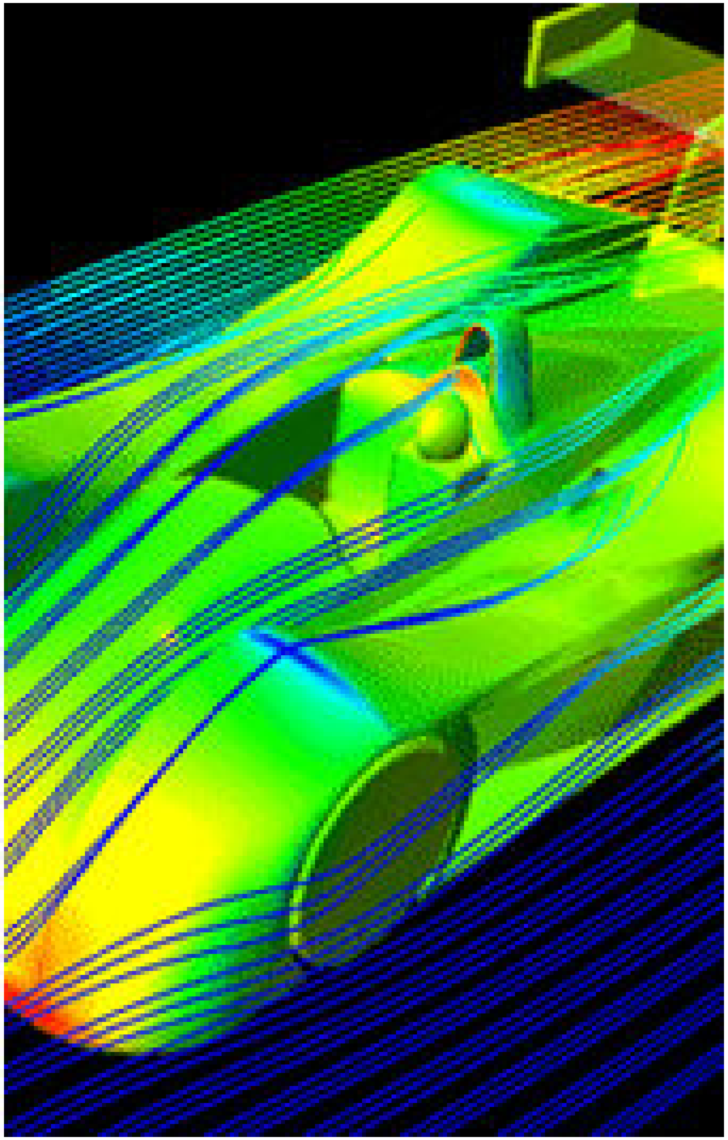


Drag ... 3

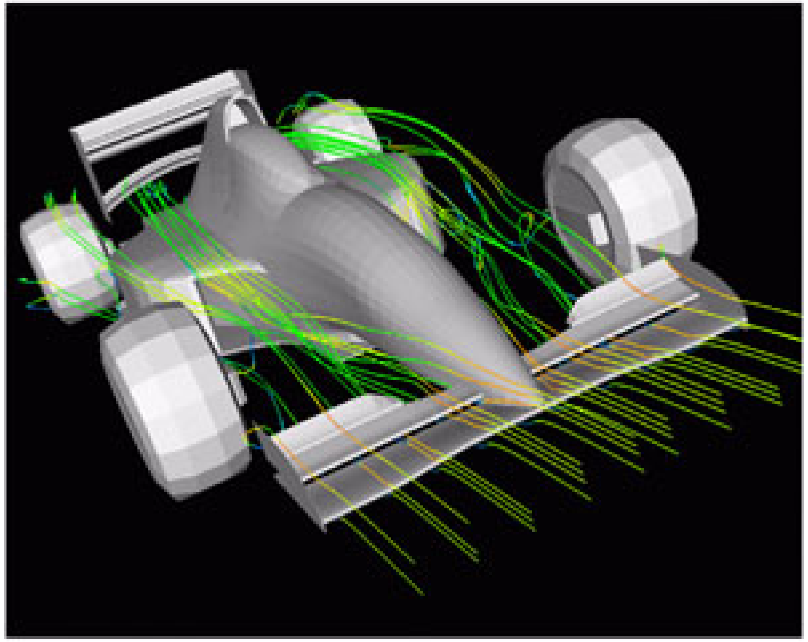
• At low Re, the flow is steady and $C_D \propto 1/\text{Re}$.
For a given body in a given fluid (D, l , ρ and η) this corresponds to $F \propto v$.

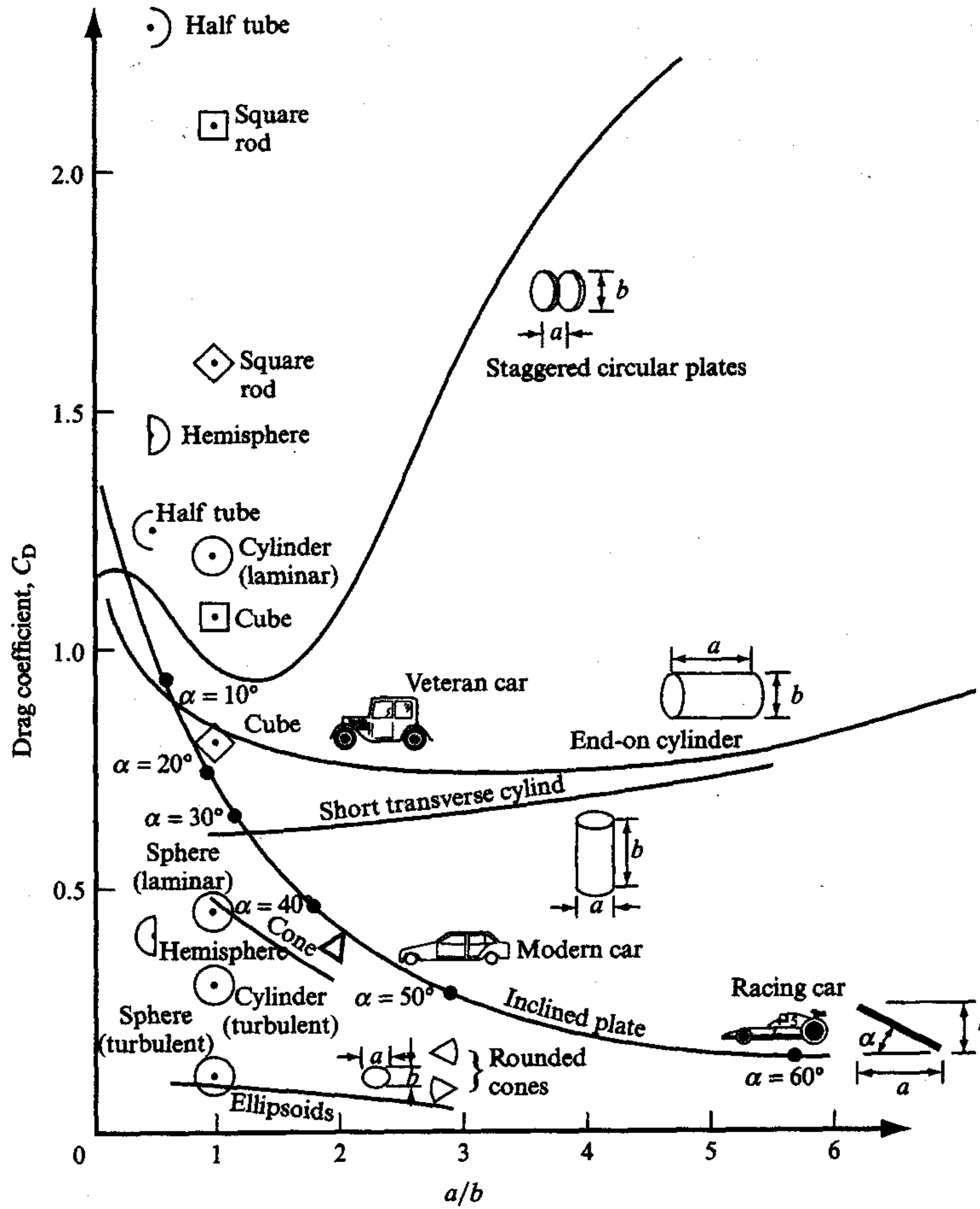
• Between Re = 10^2 to 3×10^5, C_D is approximately constant, corresponding to $F \propto v^2$.

• Above Re ~ 3×10^5, C_D drops dramatically indicating a region where an increase in v produces a decrease in F.
This Re corresponds to the onset of **turbulance** in the boundary layer resulting in a narrower wake and hence less momentum extraction from the flow (and lower F).
A full understanding of this effect includes consideration of the changes in the pressure distribution over the surface.



Le Mans car simulation





Half tube
Square rod
Staggered circular plates
Square rod
Hemisphere
Half tube
Cylinder (laminar)
Cube
Veteran car
End-on cylinder
α = 10°
Cube
α = 20°
α = 30°
Short transverse cylind
Sphere (laminar)
α = 40°
Cone
Hemisphere
Modern car
Cylinder (turbulent)
α = 50°
Inclined plate
Racing car
Sphere (turbulent)
Rounded cones
α = 60°
Ellipsoids
Drag coefficient, C_D
a/b
0
1
2
3
4
5
6
0.5
1.0
1.5
2.0

<u>Video clip (7.5) Wind Tunnel Train Model:</u> To maintain exact similarity between model and prototype flows in a wind tunnel, the model and prototype Reynolds numbers must be equal. This is usually not possible to achieve. Fortunately, flow characteristics are often not strongly influenced by the Reynolds number over the range of interest.

The flow past a train with a cross wind is studied by observing the flow of smoke injected near a model in a wind tunnel. Although the model Reynolds number is less than that for the prototype, the complex flow characteristics observed for the model should also occur for the prototype.

Video clip (9.8) Drag on a Truck: A significant portion of the power needed to drive a vehicle at highway speeds is lost overcoming aerodynamic drag. Appropriate design of the vehicle can reduce the drag and thus increase the fuel economy. Because of geometric constraints, little redesign can be done to the rear of a truck to reduce the drag. On the other hand, a simple, well-designed air deflector on the cab can help smooth the airflow past the front of the truck and reduce its drag. However, even with the deflector, there is flow separation at the top leading edge of the trailer portion of the truck.

<u>Video clip (9.9): Wing Tip Vorticies:</u> The pressure difference between the upper and lower surfaces of an airfoil causes trailing vorticies to form at the tips of the wing. The spoiler (wing) on a race car is used to produce a downward force, allowing the car to corner better. The high pressure surface for this negative lift device is the upper surface. The resulting trailing vorticies are made visible by the injection of smoke. Some airplane wings have vertical winglets at the wing tips to help reduce the effect of the trailing vorticies and therefore make the wing more efficient.

TMAA SCAN

F-4 Phantom II Caught Breaking the Sound Barrier.

Using a 35mm camera, a telephoto lens and ASA 400 film, Pat Maloney, an engineering planner, photographed an F-4 Phantom II at the moment it broke the sound barrier at the Annual Point Magu Naval Air Station Air Show. "The photograph of the visible shock wave is rare." stated Maloney. "It required a humid day, split second timing and no small measure of luck." Maloney frequently practices photography at the many air shows he attends.