Civil Engineering Department

Fluid Mechanics-II

(CV0415)

UNIT-II

Boundary Layer Concept

Prepared by: Prof. Nirali Padhiyar

Introduction

External flows past objects encompass an extremely wide variety of fluid mechanics phenomena. Clearly the character of the flow field is a function of the shape of the body.

For a given shaped object, the characteristics of the flow depend very strongly on various parameters such as size, orientation, speed, and fluid properties.

According to dimensional analysis arguments, the character of the flow should depend on the various dimensionless parameters involved.

For typical external flows the most important of these parameters are the Reynolds number, $Re = UL/\nu$, where *L*– is characteristic dimension of the body.

Introduction

For many high-Reynolds-number flows the flow field may be divided into two region

i. A viscous boundary layer adjacent to the surface
ii. The essentially inviscid flow outside the boundary layer

W know that fluids adhere the solid walls and they take the solid wall velocity. When the wall does not move also the velocity of fluid on the wall is zero.

In region near the wall the velocity of fluid particles increases from a value of zero at the wall to the value that corresponds to the external ”frictionless” flow outside the boundary layer

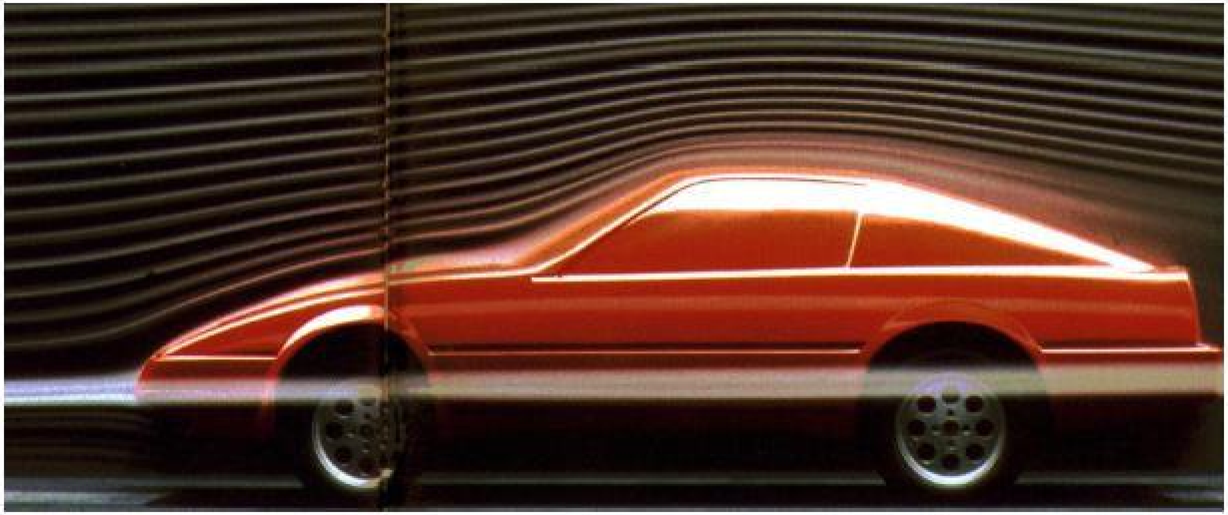


Figure 6.1: Visualization of the flow around the car. It is visible the thin layer along the body cause by viscosity of the fluid. The flow outside the narrow region near the solid boundary can be considered as ideal (inviscid).

The concept of boundary layer was first introduced by a German engineer, Prandtl in 1904.

According to Prandtl theory, when a real fluid flows past a stationary solid boundary at large values of the Reynolds number, the flow will be divided into two regions.

i. A thin layer adjoining the solid boundary, called the boundary layer, where the viscous effects and rotation cannot be neglected.

ii. An outer region away from the surface of the object where the viscous effects are very small and can be neglected. The flow behavior is similar to the upstream flow. In this case a potential flow can be assumed.

Since the fluid at the boundaries has zero velocity, there is a steep velocity gradient from the boundary into the flow. This velocity gradient in a real fluid sets up shear forces near the boundary that reduce the flow speed to that of the boundary.

That fluid layer which has had its velocity affected by the boundary shear is called ***the boundary layer.***

For smooth upstream boundaries the boundary layer starts out as a ***laminar boundary layer*** in which the fluid particles move in smooth layers.

As the laminar boundary layer increases in thickness, it becomes unstable and finally transforms into a ***turbulent boundary layer*** in which the fluid particles move in haphazard paths.

When the boundary layer has become turbulent, there is still a very thon layer next to the boundary layer that has laminar motion. It is called the ***laminar sublayer.***

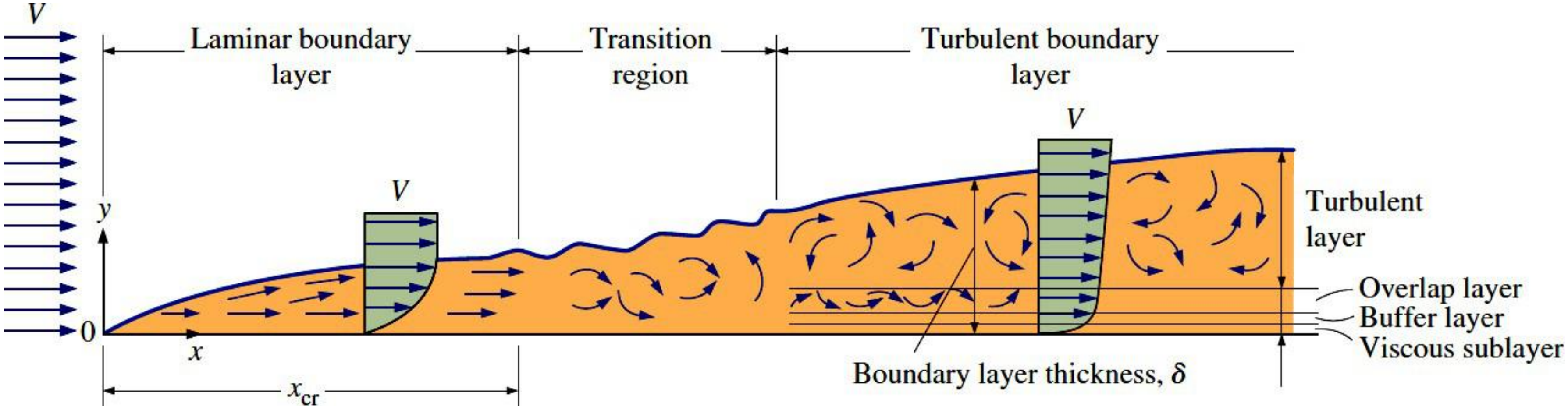


Fig. 6.2 The development of the boundary layer for flow over a flat plate, and the different flow regimes. The vertical scale has been greatly exaggerated and horizontal scale has been shortened.

The turbulent boundary layer can be considered to consist of four regions, characterized by the distance from the wall.

The very thin layer next to the wall where viscous effects are dominant is the **viscous sublayer.** The velocity profile in this layer is very nearly *linear*, and the flow is nearly parallel.

Next to the viscous sublayer is the **buffer layer,** in which turbulent effects are becoming significant, but the flow is still dominated by viscous effects.

Above the buffer layer is the **overlap layer,** in which the turbulent effects are much more significant, but still not dominant.

Above that is the **turbulent** (or **outer**) **layer** in which turbulent effects dominate over viscous effects.

Boundary layer thickness, δ

The boundary layer thickness is defined as the vertical distance from a flat plate to a point where the flow velocity reaches 99 per cent of the velocity of the free stream.

Another definition of boundary layer are the

Boundary layer displacement thickness, δ^*

Boundary layer momentum thickness, θ

Boundary layer displacement thickness, δ*

Consider two types of fluid flow past a stationary horizontal plate with velocity U as shown in Fig. 6.3. Since there is no viscosity for the case of ideal fluid (Fig. 6.3a), a uniform velocity profile is developed above the solid wall.

However, the velocity gradient is developed in the boundary layer region for the case of real fluid with the presence of viscosity and no-slip at the wall (Fig. 6.3b).

Boundary layer displacement thickness, δ*

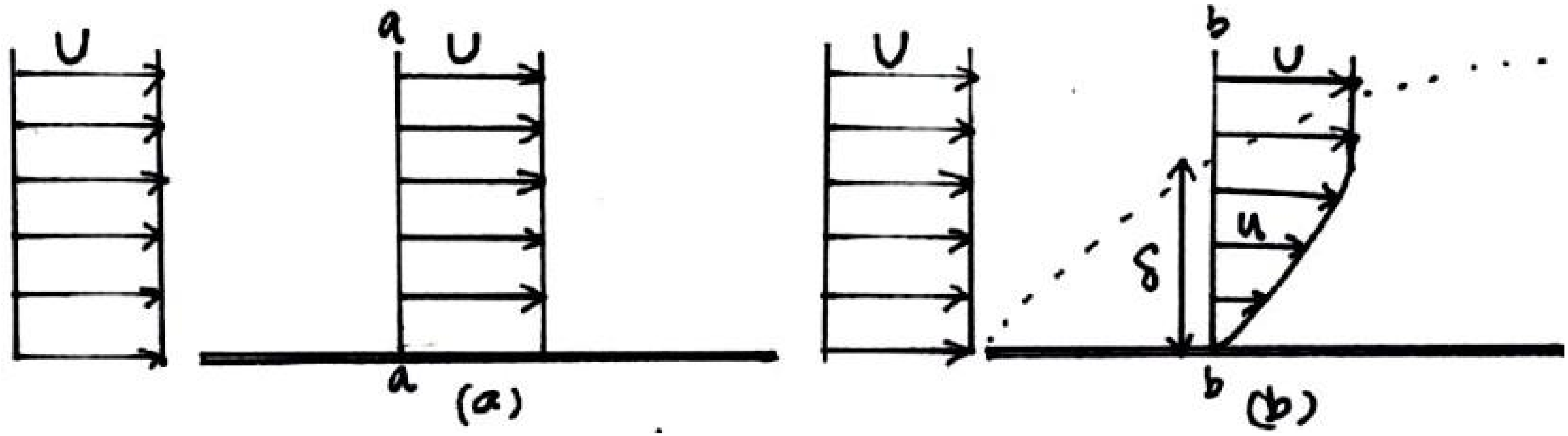


Figure 6.3 Flow over a horizontal solid surface for the case of (a) Ideal fluid (b) Real fluid

The velocity deficits through the element strip of cross section b-b is U - u . Then the reduction of mass flow rate is obtained as $\rho(U-u)bdy$ where b is the plate width.

The total mass reduction due to the presence of viscosity compared to the case of ideal fluid

$$\int_0^\delta \rho(U-u)bdy \qquad (6.1)$$

Boundary layer displacement thickness, δ*

However, if we displace the plate upward by a δ^* distance at section a-a to give mass reduction of $\rho U b \delta^*$, then the deficit of flow rates for the both cases will be identical

$$\int_0^\delta \rho(U-u)b\,dy = \rho U b \delta^*$$

and

$$\delta^* = \int_0^\delta \left(1 - \frac{u}{U}\right) dy \qquad (6.2)$$

Here δ^* is known as the boundary layer displacement thickness.

Boundary layer displacement thickness, δ*

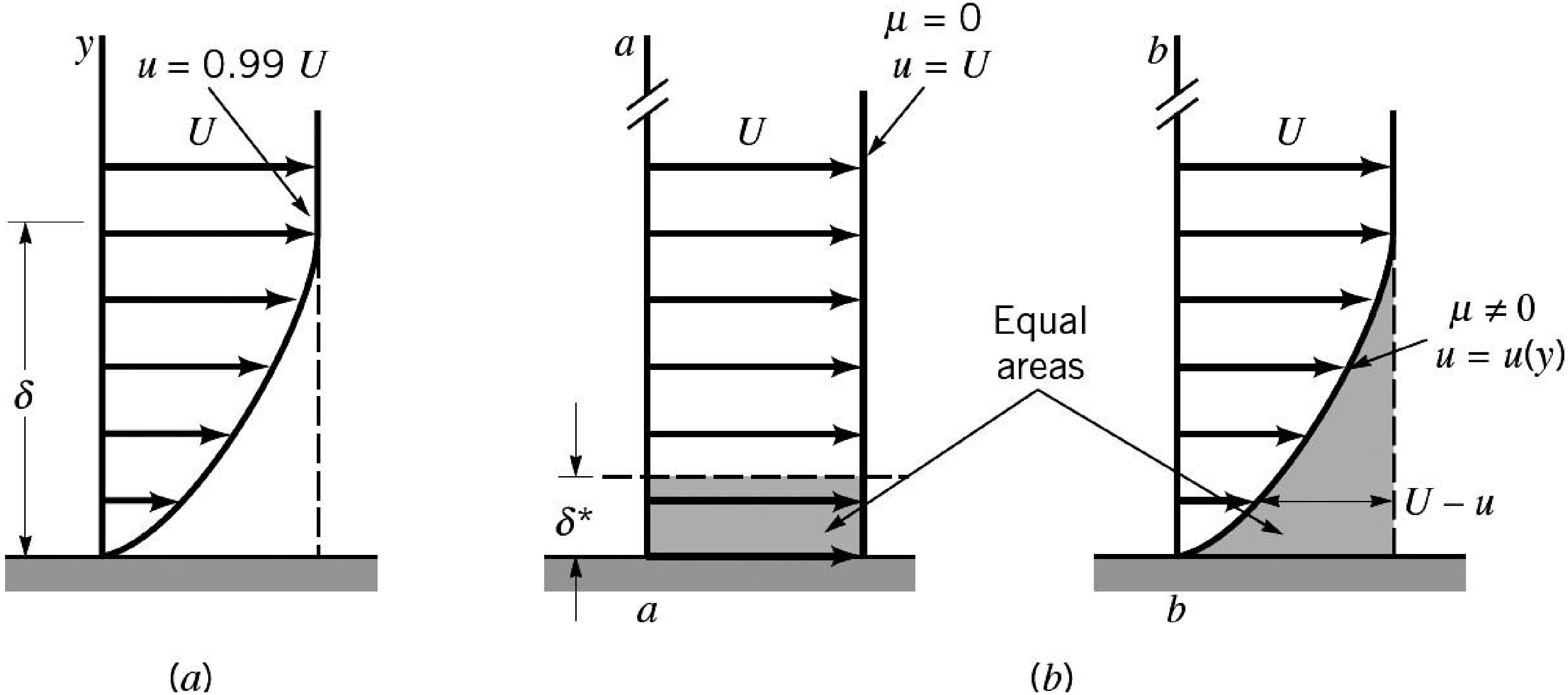


Figure 6.4: Definition of boundary layer thickness:(a) standard boundary layer(*u = 99%U),(b)* boundary layer displacement thickness .

Boundary layer displacement thickness, δ*

The displacement thickness represents the vertical distance that the solid boundary must be displaced upward so that the ideal fluid has the same mass flow rate as the real fluid.

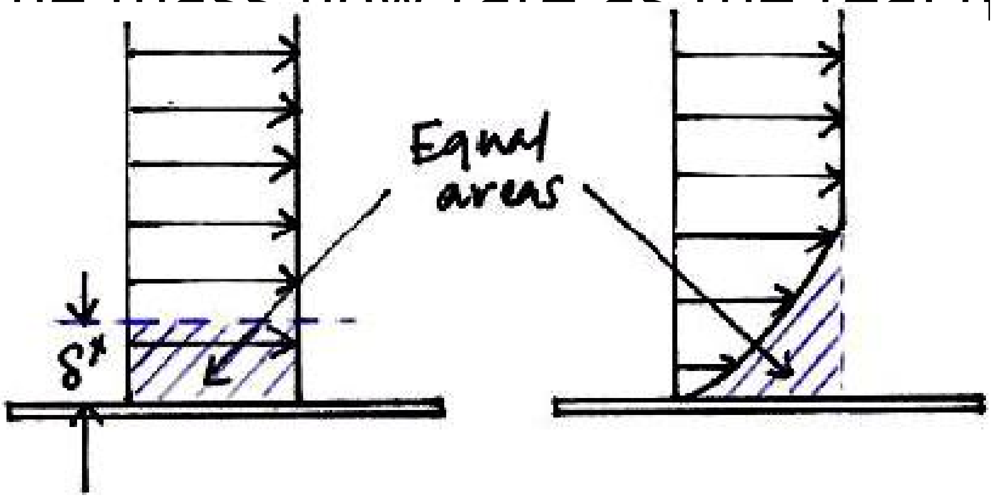


Figure 6.5 boundary layer displacement thickness

Boundary layer momentum thickness, θ

Another definition of boundary layer thickness, the boundary layer momentum thickness θ, is often used to predict the drag force on the object surface.

By referring to Fig. 6.3, again the velocity deficit through the element strip of cross section b-b contributes to deficit in momentum flux as

$$\rho u(U-u)bdy$$

(6.3)

Thus, the total momentum reductions

$$\int_0^{\delta} \rho u(U-u)bdy$$

However, if we displace the plate upward by a distance θ at section a-a to give momentum reduction of $\rho U^2 b\theta$, then the momentum deficit for the both cases will be identical if

Boundary layer momentum thickness, θ

$$\int_0^{\delta} \rho u (U - u) b\,dy = \rho U^2 b \theta$$

and

$$\theta = \int_0^{\delta} \frac{u}{U}\left(1 - \frac{u}{U}\right) dy \qquad (6.4)$$

Here, θ is known as the boundary layer momentum thickness.

The momentum thickness represents the vertical distance that the solid boundary must be displaced upward so that the ideal fluid has the same mass momentum as the real fluid

Reynolds Number and Geometry Effects

The technique of boundary layer (BL) analysis can be used to compute viscous effects near solid walls and to "patch" these onto the outer inviscid motion.

This patching is more successful as the body Reynolds number becomes larger, as shown in Fig. 6.6.

In Fig. 6.6 a uniform stream *U* moves parallel to a sharp flat plate of length *L*. If the Reynolds number UL/ν is low *(Fig. 6.6a)*, the viscous region is very broad and extends far ahead and to the sides of the plate. The plate retards the oncoming stream greatly, and small changes in flow parameters cause large changes in the pressure distribution along the plate.

There is no existing simple theory for external flow analysis at Reynolds numbers from 1 to about 1000. Such thick-shear-layer flows are typically studied by experiment or by numerical modeling of the flow field on a computer

Reynolds Number and Geometry Effects

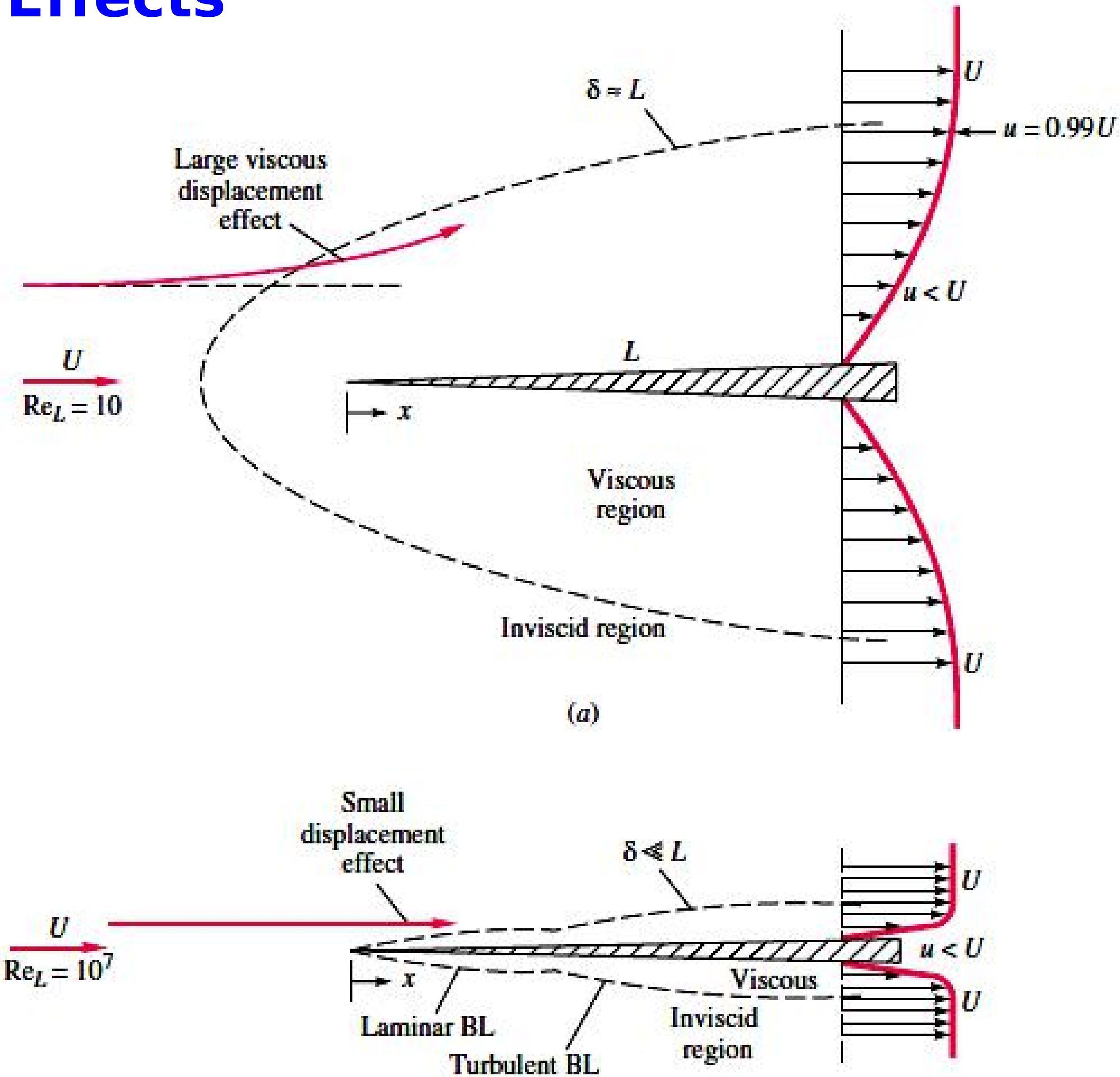


Fig. 6.6. Comparison of flow past a sharp flat plate at low and high Reynolds numbers: (*a*) *laminar,* low-Re flow; (*b*) *high-Re flow.*

Reynolds Number and Geometry Effects

A high-Reynolds-number flow (Fig. 6.6*b)* is much more amenable to boundary layer patching, as first pointed out by Prandtl in 1904.

The viscous layers, either laminar or turbulent, are very thin, thinner even than the drawing shows.

We define the boundary layer thickness δ as the locus of points where the velocity *u* parallel to the plate reaches 99 percent of the external velocity *U.*

The accepted formulas for flat-plate flow, and their approximate ranges, are

$$\frac{\delta}{x} \approx \begin{cases} \dfrac{5.0}{\mathrm{Re}_x^{1/2}} & \text{laminar} \quad 10^3 < \mathrm{Re}_x < 10^6 \\ \dfrac{0.16}{\mathrm{Re}_x^{1/7}} & \text{turbulent} \quad 10^6 < \mathrm{Re}_x \end{cases} \tag{6.5}$$

.

Reynolds Number and Geometry Effects

where $Re_x = Ux/\nu$ is called the *local Reynolds number* of the flow along the plate surface. The turbulent flow formula applies for Re_x greater than approximately 10^6 .

Some computed values are shown below

Re_x	10^4	10^5	10^6	10^7	10^8
$(\delta/x)_{lam}$	0.050	0.016	0.005		
$(\delta/x)_{turb}$			0.022	0.016	0.011

The blanks indicate that the formula is not applicable. In all cases these boundary layers are so thin that their displacement effect on the outer inviscid layer is negligible.

Thus the pressure distribution along the plate can be computed from inviscid theory as if the boundary layer were not even there.

Example 1

A long, thin flat plate is placed parallel to a 20-ft/s stream of water at 68F. At what distance x from the leading edge will the boundary layer thickness be 1 in?

Solution

Approach: Guess laminar flow first. If contradictory, try turbulent flow.

Property values: From Table for water at 68F, ν = *1.082E-5 ft²/s.*

Solution step 1: With δ = 1 in = 1/12 ft, try laminar flow

$$\frac{\delta}{x}\Big|_{\text{lam}} = \frac{5}{(Ux/\nu)^{1/2}} \quad \text{or} \quad \frac{1/12\ \text{ft}}{x} = \frac{5}{[(20\ \text{ft/s})x/(1.082\text{E-5 ft}^2/\text{s})]^{1/2}}$$

Solve for $x \approx 513$ ft

Pretty long plate! This does not sound right. Check the local Reynolds number:

$$\text{Re}_x = \frac{Ux}{\nu} = \frac{(20\ \text{ft/s})(513\ \text{ft})}{1.082\text{E-5 ft}^2/\text{s}} = 9.5\text{E8} \qquad (!)$$

Example 1

This is impossible, since laminar boundary layer flow only persists up to about 10^6 (or, with special care to avoid disturbances, up to
3 x 10^6).

Solution step 2: Try turbulent flow

$$\frac{\delta}{x} = \frac{0.16}{(Ux/\nu)^{1/7}} \quad \text{or} \quad \frac{1/12\ \text{ft}}{x} = \frac{0.16}{[(20\ \text{ft/s})x/(1.082\text{E-5 ft}^2/\text{s})]^{1/7}}$$

$$\text{Solve for} \quad x \approx 5.17\ \text{ft}$$

Check Re_x = (20 ft/s)(5.17 ft)/(1.082E-5 ft^2/s) = 9.6E6 > 10^6. OK, turbulent flow.

Boundary Layer: Momentum Integral Estimates

A shear layer of unknown thickness grows along the sharp flat plate in Fig. 6.7. The no-slip wall condition retards the flow, making it into a rounded profile *u(x,y),* which merges into the external velocity *U* = constant at a "thickness" y = δ(x).

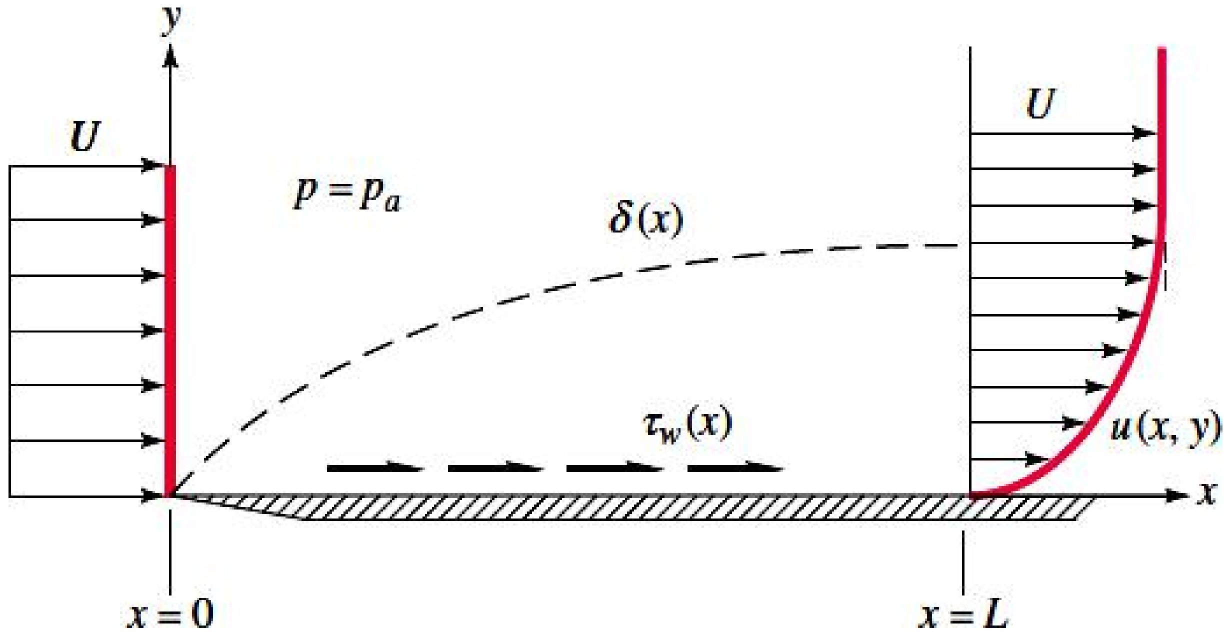


Fig. 6.7 Growth of a boundary layer on a flat plate.

Boundary Layer: Momentum Integral Estimates

The drag force on the plate is given by the following momentum integral across the exit plane:

$$D(x) = \rho b \int_0^{\delta(x)} u(U - u)\, dy \qquad (6.6)$$

where b is the plate width into the paper and the integration is carried out along a vertical plane x = constant.

Equation (6.6) was derived in 1921 by Kármán, who wrote it in the convenient form of the *momentum thickness as:*

$$D(x) = \rho b U^2 \theta \qquad \theta = \int_0^{\delta} \frac{u}{U}\left(1 - \frac{u}{U}\right) dy \qquad (6.7)$$

Momentum thickness is a measure of total plate drag which also equals the integrated wall shear stress along the plate:

Boundary Layer: Momentum Integral Estimates

$$D(x) = b\int_0^x \tau_w(x)\,dx$$

or

$$\frac{dD}{dx} = b\tau_w \qquad (6.8)$$

Meanwhile, the derivative of Eq. (6.7), with *U = constant, is*

$$\frac{dD}{dx} = \rho b U^2 \frac{d\theta}{dx}$$

By comparing this with eq. (6.8), the momentum integral relation for flat-plate boundary layer flow is given by

$$\tau_w = \rho U^2 \frac{d\theta}{dx} \qquad (6.9)$$

It is valid for either laminar or turbulent flat-plate flow.

Boundary Layer: Momentum Integral Estimates

To get a numerical result for laminar flow, assuming that the velocity profiles have an approximately parabolic shape

$$u(x, y) \approx U\left(\frac{2y}{\delta} - \frac{y^2}{\delta^2}\right) \qquad 0 \le y \le \delta(x) \tag{6.10}$$

which makes it possible to estimate both momentum thickness and wall shear:

$$\theta = \int_0^{\delta}\left(\frac{2y}{\delta} - \frac{y^2}{\delta^2}\right)\left(1 - \frac{2y}{\delta} + \frac{y^2}{\delta^2}\right)dy \approx \frac{2}{15}\delta$$

$$\tau_w = \mu\frac{\partial u}{\partial y}\bigg|_{y=0} \approx \frac{2\mu U}{\delta} \tag{6.11}$$

By substituting these values into the momentum integral relation (eq. (6.9) and rearranging we obtain

$$\delta\, d\delta \approx 15\,\frac{\nu}{U}\,dx \tag{6.12}$$

Boundary Layer: Momentum Integral Estimates

where ν = *μ* /*ρ*. We can integrate from 0 to x, assuming that δ = 0 at x = 0, the leading edge

$$\frac{1}{2}\delta^2 = \frac{15\nu x}{U}$$

or

$$\frac{\delta}{x} \approx 5.5\left(\frac{\nu}{Ux}\right)^{1/2} = \frac{5.5}{\mathrm{Re}_x^{1/2}} \qquad (6.13)$$

This is the desired thickness estimate. It is only 10 percent higher than the known accepted solution for laminar flat-plate flow (eq. (6.5)).

We can also obtain a shear stress estimate along the plate from the above relations

$$c_f = \frac{2\tau_w}{\rho U^2} \approx \left(\frac{\frac{8}{15}}{\mathrm{Re}_x}\right)^{1/2} = \frac{0.73}{\mathrm{Re}_x^{1/2}} \qquad (6.14)$$

Boundary Layer: Momentum Integral Estimates

This is only 10 percent higher than the known exact laminar-plate-flow solution $c_f = 0.664/Re_x^{1/2}$

The dimensionless quantity c_f, called the *skin friction coefficient, is* analogous to the friction factor *f* in ducts.

A boundary layer can be judged as “thin” if, say, the ratio δ/x *is* less than about 0.1. This occurs at $\delta/x = 0.1 = 5.0/Re_x^{1/2}$ *or at* $Re_x = 2500$.

For Re_x less than 2500 we can estimate that boundary layer theory fails because the thick layer has a significant effect on the outer inviscid flow.

The upper limit on Re_x for *laminar flow* is about 3×10^6, where measurements on a smooth flat plate show that the flow undergoes transition to a turbulent boundary layer.

From 3×10^6 upward the turbulent Reynolds number may be arbitrarily large, and a practical limit at present is 5×10^{10} for oil supertankers

Boundary Layer: Momentum Integral Estimates

For parallel flow over a flat plate, the pressure drag is zero, and thus the drag coefficient is equal to the *friction drag coefficient,* or simply the *friction coefficient*).

Once the average friction coefficient C_f is available, the drag (or friction) force over the surface is determined from

$$F_D = F_f = \frac{1}{2} C_f A \rho V^2$$

where *A* is the surface area of the plate exposed to fluid flow. When both sides of a thin plate are subjected to flow, *A* becomes the total area of the top and bottom surfaces.

Example 2

Are low-speed, small-scale air and water boundary layers really thin? Consider flow at *U* =1 ft/s past a flat plate 1 ft long.
Compute the boundary layer thickness at the trailing edge for (a) air and (b) water at 68F.

Solution

From Table ν_{air} = 1.61 E-4 ft²/s. The trailing-edge Reynolds number thus is

$$Re_L = \frac{UL}{\nu} = \frac{(1 \text{ ft/s})(1 \text{ ft})}{1.61 \text{ E-4 ft}^2/\text{s}} = 6200$$

Since this is less than 10^6, the flow is presumed laminar, and since it is greater than 2500, the boundary layer is reasonably thin. The predicted laminar thickness is

Example 2

$$\frac{\delta}{x} = \frac{5.0}{\sqrt{6200}} = 0.0634$$

or, at $x = 1$ ft, $\delta = 0.0634 \text{ ft} \approx 0.76 \text{ in}$

From Table ν_{water} = 1.08 E-5 ft²/s. The trailing-edge Reynolds number is

$$\text{Re}_L = \frac{(1 \text{ ft/s})(1 \text{ ft})}{1.08 \text{ E-5 ft}^2\text{/s}} \approx 92{,}600$$

This again satisfies the laminar and thinness conditions.

The boundary layer thickness is

$$\frac{\delta}{x} \approx \frac{5.0}{\sqrt{92{,}600}} = 0.0164$$

or, at $x = 1$ ft, $\delta = 0.0164 \text{ ft} \approx 0.20 \text{ in}$

Part I

Laminar and Turbulent Pipe Flow

Introduction

Fluid flow in circular and noncircular pipes is commonly encountered in practice.

The hot and cold water that we use in our homes is pumped through pipes. Water in a city is distributed by extensive piping networks. Oil and natural gas are transported hundreds of miles by large pipelines. Blood is carried throughout our bodies by arteries and veins. The cooling water in an engine is transported by hoses to the pipes in the radiator where it is cooled as it flows.

The fluid in such applications is usually forced to flow by afan or pump through a flow section.

We pay particular attention to friction, which is directly related to the pressure drop and head loss during flow through pipes and ducts.

The pressure drop is then used to determine the pumping power requirement.

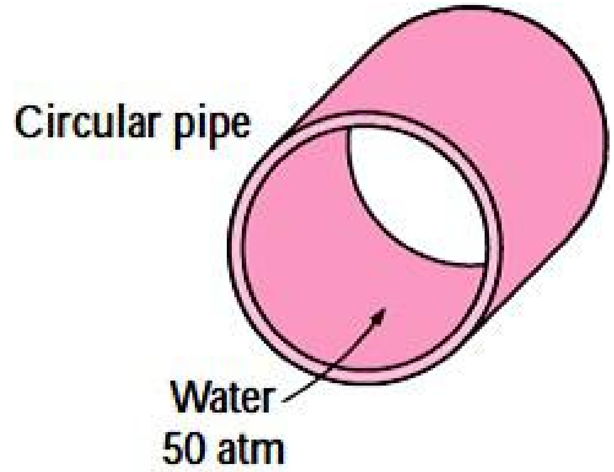
A typical piping system involves pipes of different diameters connected to each other by various fittings or elbows to route the fluid, valves to control the flow rate, and pumps to pressurize the fluid.

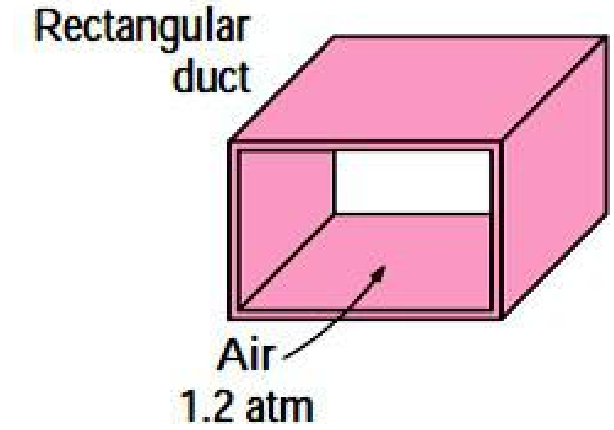
The terms ***pipe***, ***duct***, *and* ***conduit*** are usually used interchangeably for flow sections.

In general, flow sections of circular cross section are referred to as pipes (especially when the fluid is a liquid), and flow sections of noncircular cross section as ducts (especially when the fluid is a gas) Small diameter pipes are usually referred to as tubes.

Most fluids, especially liquids, are transported in circular pipes. This is because pipes with a circular cross section can withstand large pressure differences between the inside and the outside without undergoing significant distortion.

Noncircular pipes are usually used in applications such as the heating and cooling systems of buildings where the pressure difference is relatively small, the manufacturing and installation costs are lower, and the available space is limited for ductwork.

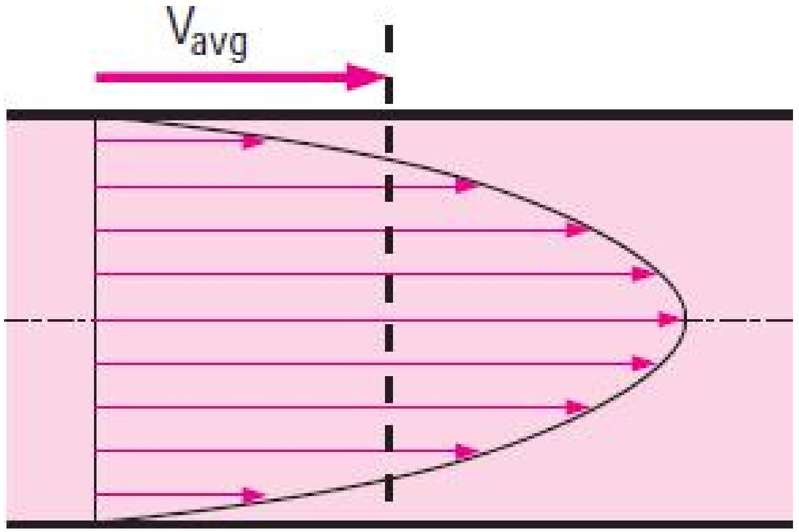




The fluid velocity in a pipe changes from zero at the surface because of the no-slip condition to a maximum at the pipe center.

In fluid flow, it is convenient to work with an average velocity V_{avg}, which remains constant in incompressible flow when the cross-sectional area of the pipe is constant.

The change in average velocity due to change in density and temperature and due to friction is usually small and is thus disregarded in calculations.



Average velocity V_{avg} is defined as the average speed through a cross section. For fully developed laminar pipe flow, V_{avg} is half of maximum velocity.

The value of the average velocity V_{avg} at some streamwise cross-section is determined from the requirement that the conservation of mass principle be satisfied. That is,

$$\dot{m} = \rho V_{avg} A_c = \int_{A_c} \rho u(r)\, dA_c$$

where $\dot{m}$ is the mass flow rate, ρ is the density, A_c is the cross-sectional area, and u(r) is the velocity profile. Then the average velocity for incompressible flow in a circular pipe of radius R can be expressed as

$$V_{avg} = \frac{\int_{A_c} \rho u(r)\, dA_c}{\rho A_c} = \frac{\int_0^R \rho u(r) 2\pi r\, dr}{\rho \pi R^2} = \frac{2}{R^2} \int_0^R u(r) r\, dr$$

Therefore, when we know the flow rate or the velocity profile, the average velocity can be determined easily.

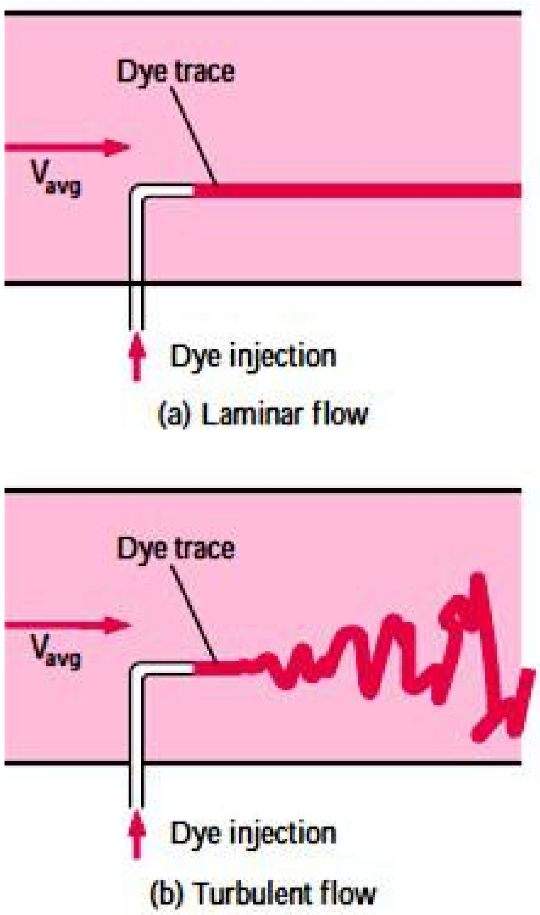
LAMINAR AND TURBULENT FLOWS

Fluid flow in a pipe is streamlined at low velocities but turns chaotic as the velocity is increased above a critical value.

A **laminar** flow is characterized **by smooth streamlines and highly ordered motion**, and **turbulent flow** is characterized by velocity fluctuations and highly disordered motion.

The **transition from laminar to turbulent flow does not occur suddenly;** rather, it occurs over some region in which the flow fluctuates between laminar and turbulent flows before it becomes fully turbulent.

Most flows encountered in practice are turbulent. Laminar flow is encountered when highly viscous fluids such as oils flow in small pipes or narrow passages.

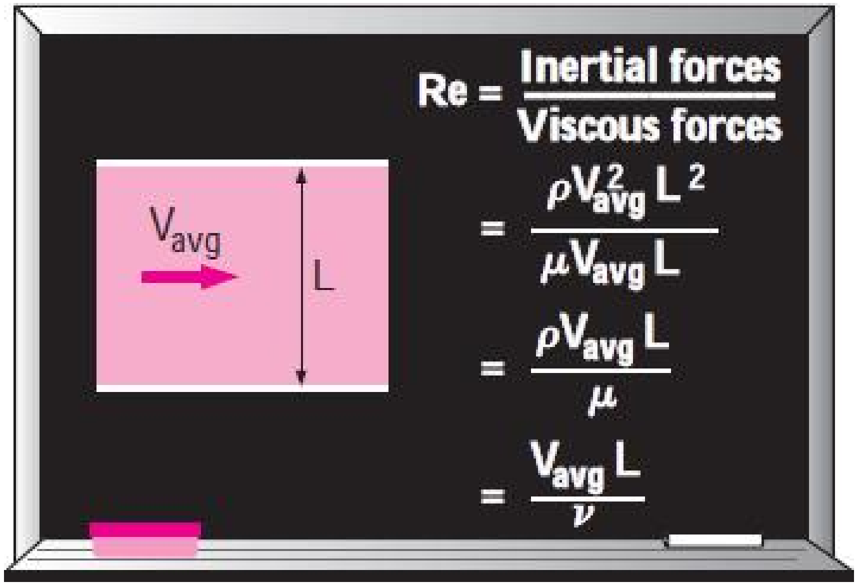


Reynolds Number

The transition from laminar to turbulent flow depends on the geometry, surface roughness, flow velocity, surface temperature, and type of fluid, among other things.

After exhaustive experiments in the 1880s, Osborne Reynolds discovered that the flow regime depends mainly on the ratio of inertial forces to viscous forces in the fluid. This ratio is called the **Reynolds number** and is expressed for internal flow in a circular pipe as

$$Re = \frac{\text{Inertial forces}}{\text{Viscous forces}} = \frac{V_{avg}D}{\nu} = \frac{\rho V_{avg}D}{\mu}$$



Reynolds Number

where V_{avg} = average flow velocity (m/s), D = characteristic length of the geometry (diameter in this case, in m), and $\nu = \mu/\rho$ = kinematic viscosity of the fluid (m^2/s).

Note that the Reynolds number is a *dimensionless quantity.*

At large Reynolds numbers, the inertial forces, which are proportional to the fluid density and the square of the fluid velocity, are large relative to the viscous forces, and thus the viscous forces cannot prevent the random and rapid fluctuations of the fluid.

At small or moderate Reynolds numbers, however, the viscous forces are large enough to suppress these fluctuations and to keep the fluid "in line."

Thus the flow is turbulent in the first case and laminar in the second.

Reynolds Number

The Reynolds number at which the flow becomes turbulent is called the **critical Reynolds number, Re_{cr} .**

The value of the critical Reynolds number is different for different geometries and flow conditions. For internal flow in a circular pipe, the generally accepted value of the critical Reynolds number is $Re_{cr} = 2300$.

For flow through noncircular pipes, the Reynolds number is based on the **hydraulic diameter D_h** defined as

$$D_h = \frac{4A_c}{p}$$

where A_c is the cross-sectional area of the pipe and p is its wetted perimeter. The hydraulic diameter is defined such that it reduces to ordinary diameter D for circular pipes,

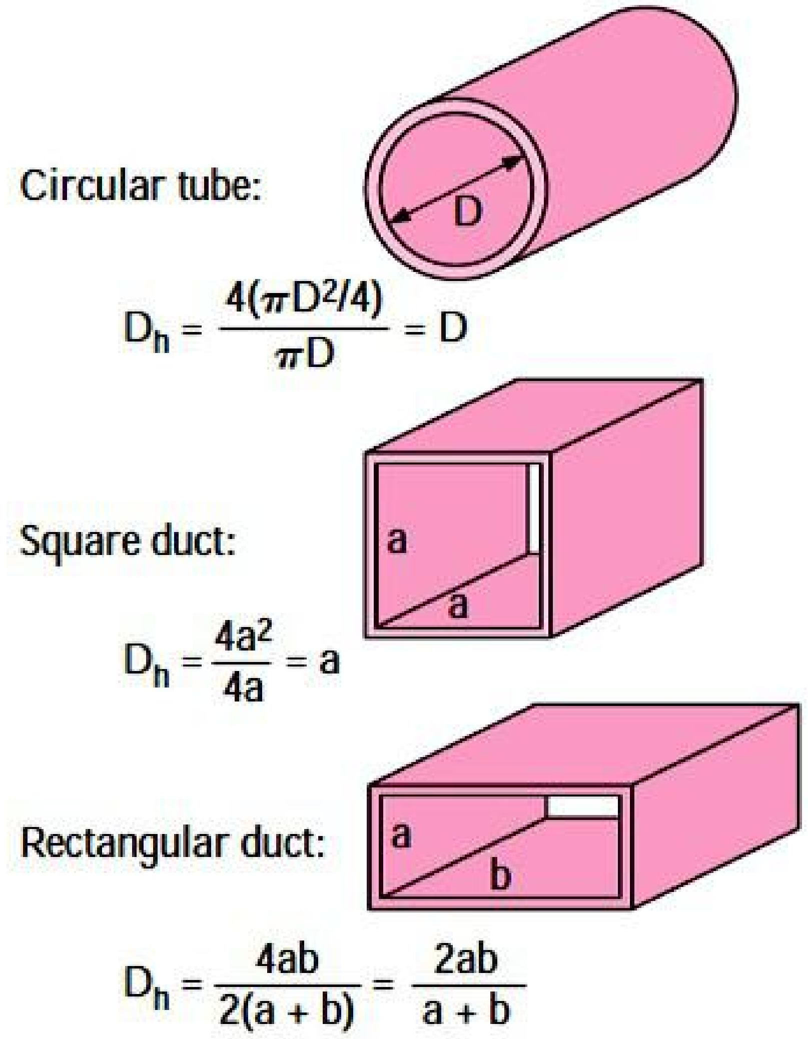
Reynolds Number

For circular pipes

$$D_h = \frac{4A_c}{p} = \frac{4(\pi D^2/4)}{\pi D} = D$$

For flow in circular pipes

$Re \lesssim 2300$	laminar flow
$2300 \lesssim Re \lesssim 4000$	transitional flow
$Re \gtrsim 4000$	turbulent flow



The Entrance Region

Consider a fluid entering a circular pipe at a uniform velocity. Because of the no-slip condition, the fluid particles in the layer in contact with the surface of the pipe come to a complete stop.

This layer also causes the fluid particles in the adjacent layers to slow down gradually as a result of friction.

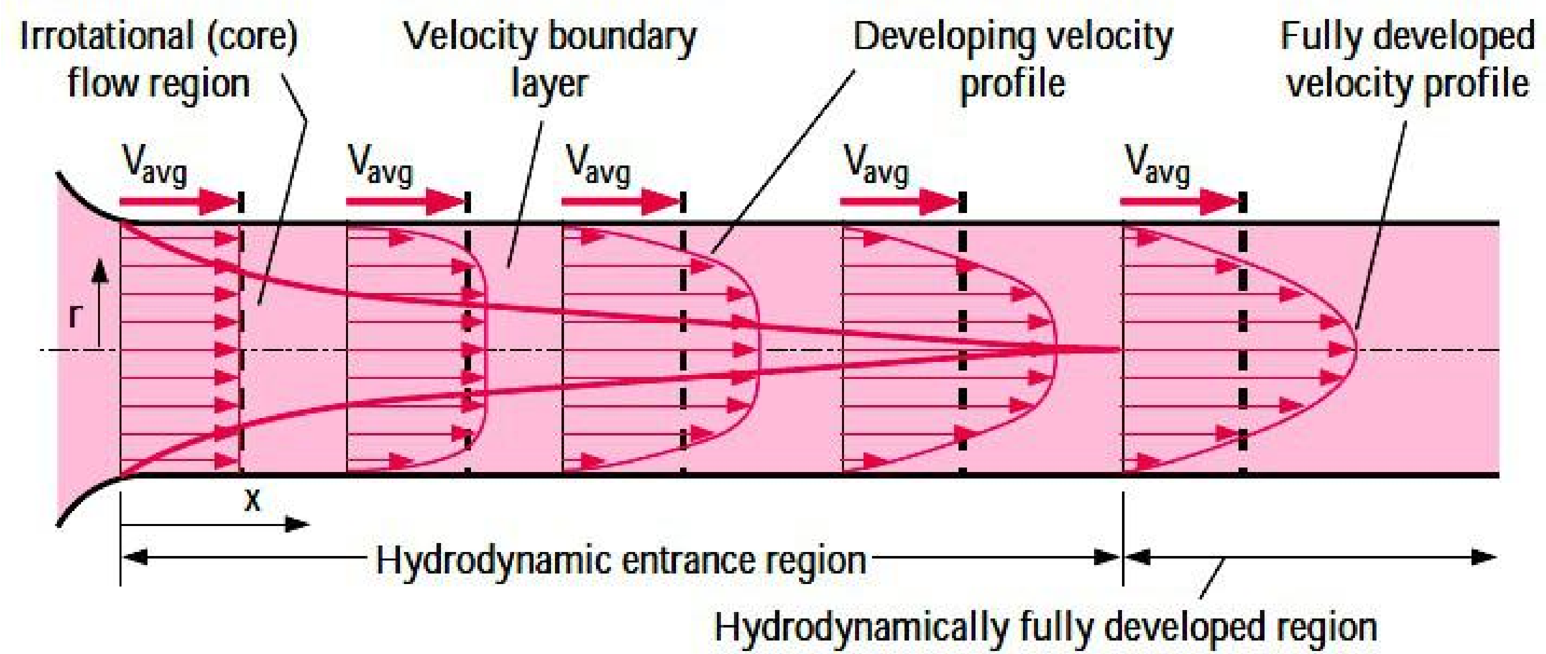
The region of the flow in which the effects of the viscous shearing forces caused by fluid viscosity are felt is called the **velocity boundary layer or** just the **boundary layer.**

The hypothetical boundary surface divides the flow in apipe into two regions: the **boundary layer region,** in which the viscous effects and the velocity changes are significant, and the **irrotational (core) flow region,** in which the **frictional effects are negligible** and the velocity remains essentially constant in the radial direction.

The Entrance Region

The thickness of this boundary layer increases in the flow direction until the boundary layer reaches the pipe center and thus fills the entire pipe.

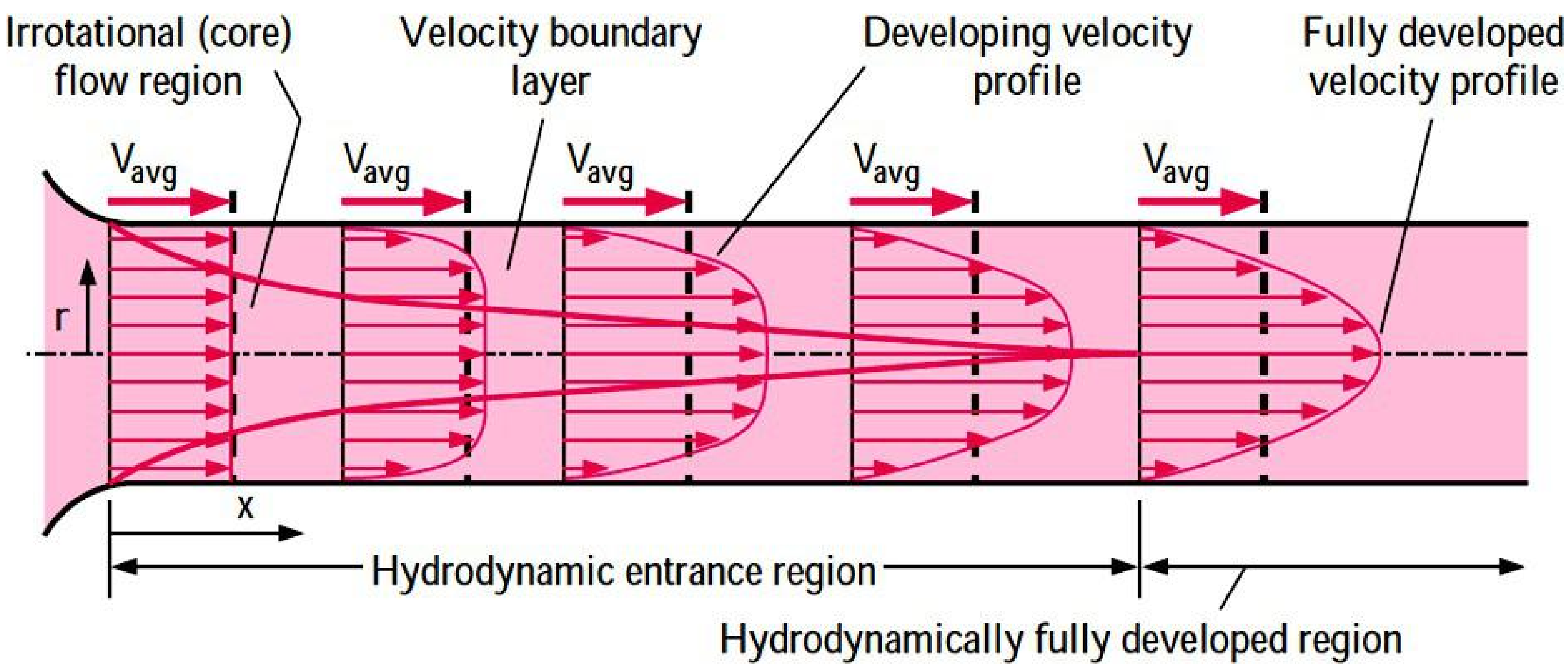
The region from the pipe inlet to the point at which the boundary layer merges at the centerline is called the **hydrodynamic entrance region,** and the length of this region is called the **hydrodynamic entry length L_h.**



The Entrance Region

Flow in the entrance region is called **hydrodynamically developing** flow since this is the region where the velocity profile develops.

The region beyond the entrance region in which the velocity profile is fully developed and remains unchanged is called the **hydrodynamically fully developed** region.



The Entrance Region

The velocity profile in the fully developed region is parabolic in laminar flow and somewhat flatter (or fuller) in turbulent flow due to eddy motion and more vigorous mixing in the radial direction.

Entry Lengths

The hydrodynamic entry length is usually taken to be the distance from the pipe entrance to where the wall shear stress (and thus the friction factor) reaches within about 2 percent of the fully developed value.

In *laminar flow,* the hydrodynamic entry length is given approximately as

$$L_{h,\ laminar} \cong 0.05ReD$$

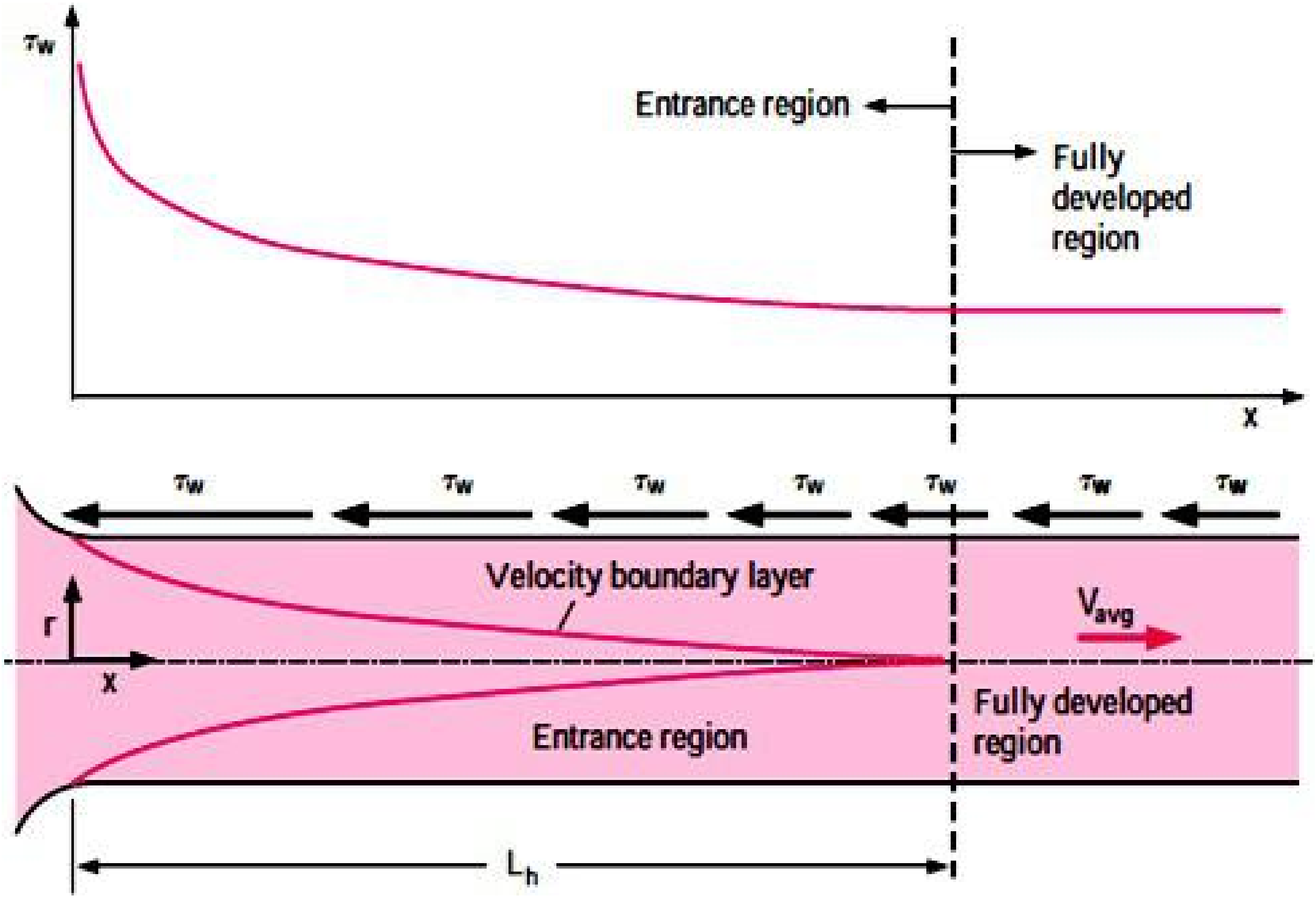


Fig. The variation of wall shear stress in the flow direction for flow in a pipe from the entrance region into the fully developed region.

The Entrance Region

In turbulent flow, the intense mixing during random fluctuations usually overshadows the effects of molecular diffusion.

The hydrodynamic entry length for turbulent flow can be approximated as [see Bhatti and Shah (1987) and Zhi-qing (1982)]

$$L_{h,\ turbulent} = 1.359 D Re_D^{1/4}$$

The entry length is much shorter in turbulent flow, as expected, and its dependence on the Reynolds number is weaker.

In many pipe flows of practical engineering interest, the entrance effects become insignificant beyond a pipe length of 10 diameters, and the hydrodynamic entry length is approximated as

The Entrance Region

$$L_{h,\ turbulent} \approx 10D$$

The pipes used in practice are usually several times the length of the entrance region, and thus the flow through the pipes is often assumed to be fully developed for the entire length of the pipe. This simplistic approach gives reasonable results for long pipes but sometimes poor results for short ones since it under predicts the wall shear stress and thus the friction factor.

Laminar Flow in Pipes

Flow in pipes is laminar for $Re \leq 2300$, and that the flow is fully developed if the pipe is sufficiently long (relative to the entry length) so that the entrance effects are negligible.

In this section we consider the steady laminar flow of an incompressible fluid with constant properties in the fully developed region of a straight circular pipe.

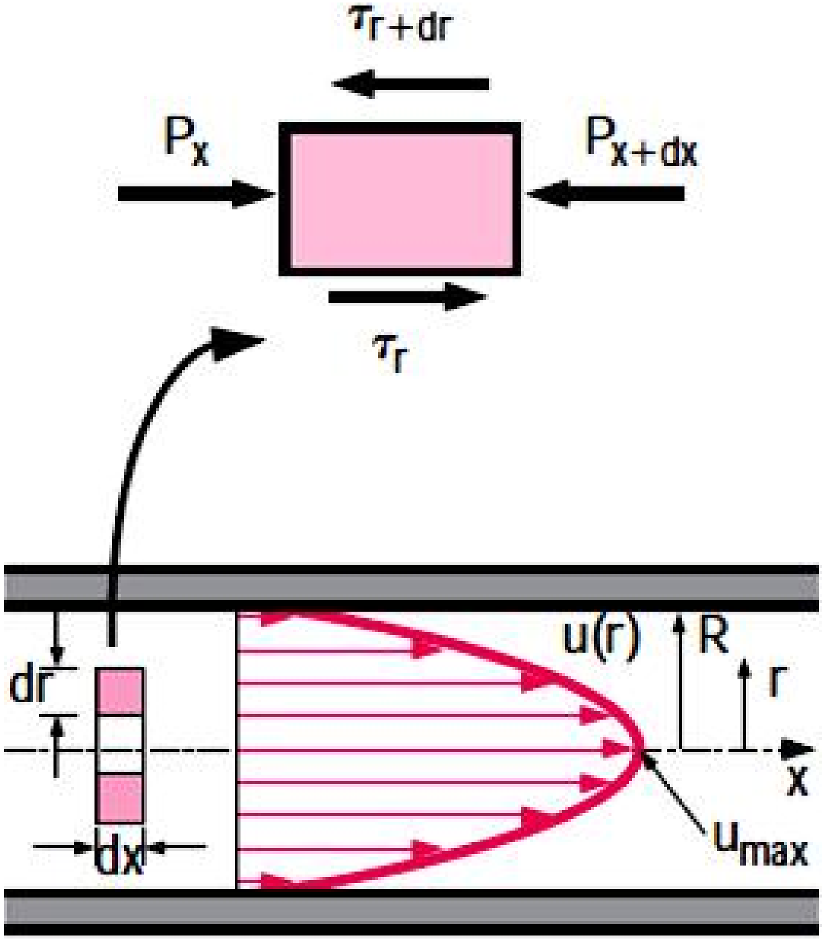
In fully developed laminar flow, each fluid particle moves at a constant axial velocity along a streamline and the velocity profile $u(r)$ remains unchanged in the flow direction. There is no motion in the radial direction, and thus the velocity component in the direction normal to flow is everywhere zero. There is no acceleration since the flow is steady and fully developed.

Laminar Flow in Pipes

Consider a ring-shaped differential volume element of radius r, thickness dr, and length dx oriented coaxially with the pipe, as shown in the Fig.

The volume element involves only pressure and viscous effects and thus the pressure and shear forces must balance each other.

The pressure force acting on a submerged plane surface is the product of the pressure at the centroid of the surface and the surface area.



Laminar Flow in Pipes

A force balance on the volume element in the flow direction gives

$$(2\pi r\,dr\,P)_x - (2\pi r\,dr\,P)_{x+dx} + (2\pi r\,dx\,\tau)_r - (2\pi r\,dx\,\tau)_{r+dr} = 0$$

which indicates that in fully developed flow in a horizontal pipe, the viscous and pressure forces balance each other.

Dividing by 2πdrdx *and* rearranging,

$$r\,\frac{P_{x+dx} - P_x}{dx} + \frac{(r\tau)_{r+dr} - (r\tau)_r}{dr} = 0$$

Taking the limit as dr, dx $\rightarrow$ 0 gives

$$r\,\frac{dP}{dx} + \frac{d(r\tau)}{dr} = 0$$

Laminar Flow in Pipes

Substituting $\tau = -\mu(du/dr)$ and taking μ = constant gives the desired equation,

$$\frac{\mu}{r}\frac{d}{dr}\left(r\frac{du}{dr}\right) = \frac{dP}{dx}$$

The quantity du/dr is negative in pipe flow, and the negative sign is included to obtain positive values for τ.
(Or, du/dr = -du/dy since y = R - r)

Rearranging and integrating twice gives

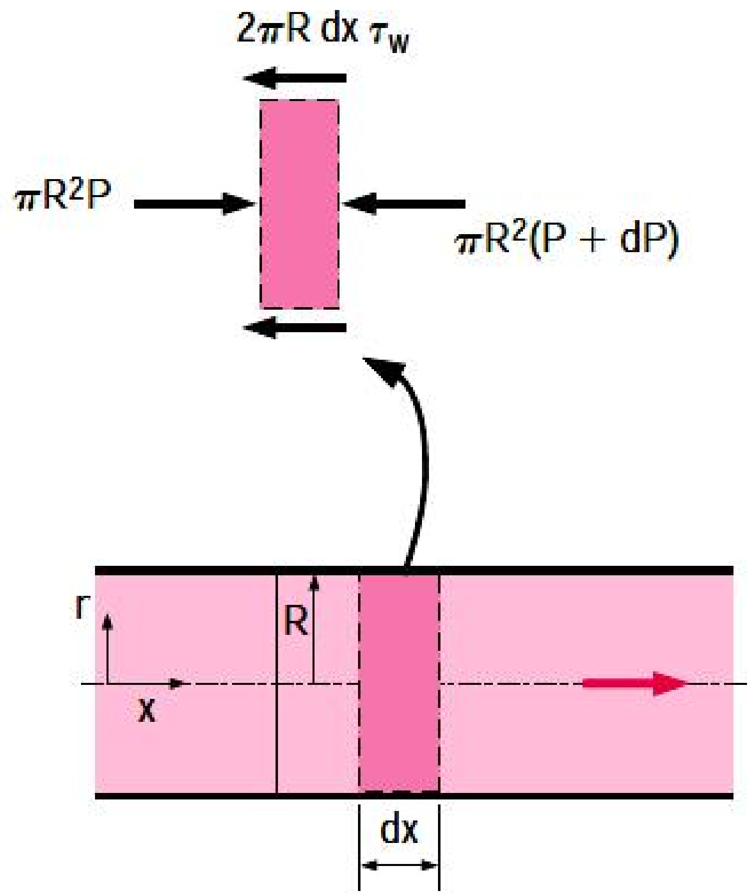
$$u(r) = \frac{1}{4\mu}\left(\frac{dP}{dx}\right) + C_1 \ln r + C_2$$

Laminar Flow in Pipes

Writing a force balance on a volume element of radius R and thickness dx (a slice of the pipe), gives $\frac{dP}{dx} = -\frac{2\tau_w}{R}$

Here τ_w is constant since the viscosity and the velocity profile are constants in the fully developed region.

Therefore, dP/dx = constant.



Force balance:

$$\pi R^2 P - \pi R^2 (P + dP) - 2\pi R\, dx\, \tau_w = 0$$

Simplifying:

$$\frac{dP}{dx} = -\frac{2\tau_w}{R}$$

Laminar Flow in Pipes

The velocity profile u(r) is obtained by applying the boundary conditions $\partial u/\partial r = 0$ at r = 0 (because of symmetry about the centerline) and u = 0 at r = R (the no-slip condition at the pipe surface). We get

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

Therefore, the velocity profile in fully developed laminar flow in a pipe is parabolic with a maximum at the centerline and minimum (zero) at the pipe wall.

Also, the axial velocity u is positive for any r, and thus the axial pressure gradient dP/dx must be negative (i.e., pressure must decrease in the flow direction because of viscous effects).

Laminar Flow in Pipes

The average velocity is determined from its definition by substituting u(r) and performing the integration. It gives

$$V_{avg} = \frac{2}{R^2}\int_0^R u(r) r\, dr = \frac{-2}{R^2}\int_0^R \frac{R^2}{4\mu}\left(\frac{dP}{dx}\right)\left(1 - \frac{r^2}{R^2}\right) r\, dr = -\frac{R^2}{8\mu}\left(\frac{dP}{dx}\right)$$

Combining the last two equations, the velocity profile is rewritten as

$$u(r) = 2V_{avg}\left(1 - \frac{r^2}{R^2}\right)$$

This is a convenient form for the velocity profile since V_{avg} can be determined easily from the flow rate information.

The maximum velocity occurs at the centerline and is determined by substituting $r = 0$,

$$u_{max} = 2V_{avg}$$

Pressure Drop and Head Loss

The average velocity for laminar flow in a horizontal pipe is

Horizontal pipe: $$V_{avg} = \frac{(P_1 - P_2)R^2}{8\mu L} = \frac{(P_1 - P_2)D^2}{32\mu L} = \frac{\Delta P\, D^2}{32\mu L}$$

Then the volume flow rate for laminar flow through a horizontal pipe of diameter *D and length L becomes*

$$\dot{V} = V_{avg} A_c = \frac{(P_1 - P_2)R^2}{8\mu L}\pi R^2 = \frac{(P_1 - P_2)\pi D^4}{128\mu L} = \frac{\Delta P\, \pi D^4}{128\mu L}$$

Turbulent Flow in Pipes

Most flows encountered in engineering practice are turbulent, and thus it is important to understand how turbulence affects wall shear stress.

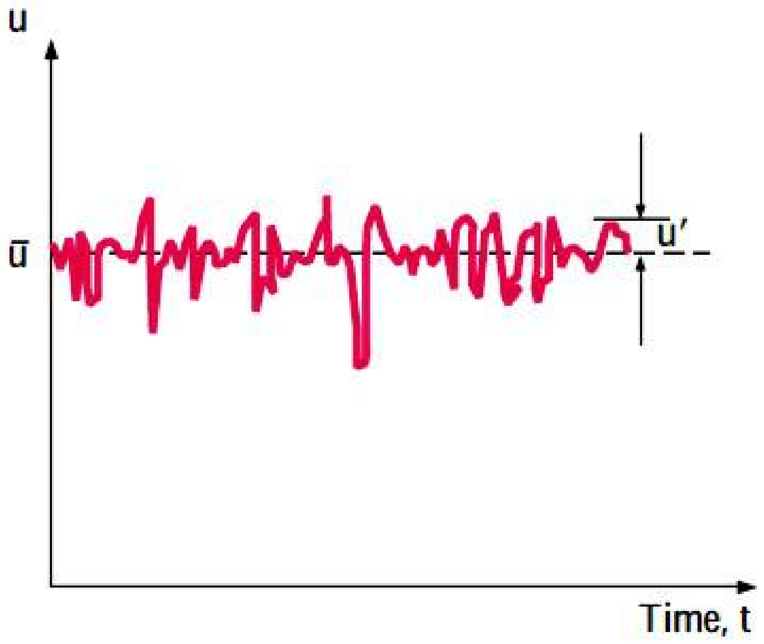
However, turbulent flow is a complex mechanism dominated by fluctuations, and despite tremendous amounts of work done in this area by researchers, the theory of turbulent flow remains largely undeveloped.

Therefore, we must rely on experiments and the empirical or semi-empirical correlations developed for various situations.

Turbulent flow is characterized by random and rapid fluctuations of swirling regions of fluid, called **eddies,** throughout the flow. These fluctuations provide an additional mechanism for momentum and energy transfer.

Turbulent Flow in Pipes

Fluctuations of the velocity component u with time at a specified location in turbulent flow shown in fig below.



Instantaneous values of the velocity fluctuate about an average value, which suggests that the velocity can be expressed as the sum of an average value $\bar{u}$ and fluctuating component u',

$$u = \bar{u} + u'$$

Turbulent Flow in Pipes

Turbulent Shear Stress

The turbulent shear stress consists of two parts: the laminar component, which accounts for the friction between layers in the flow direction (expressed as $\tau_{lam} = -\mu \; d\bar{u}/dr)$, and the turbulent component, w r the friction between the fluctuating fluid particles and the fluid body (denoted as τ_{turb} and is related to the fluctuation components of velocity).

Then the total shear stress in turbulent flow can be expressed as

$$\tau_{total} = \tau_{lam} + \tau_{turb}$$

Turbulent Flow in Pipes

The total shear stress can be expressed conveniently as

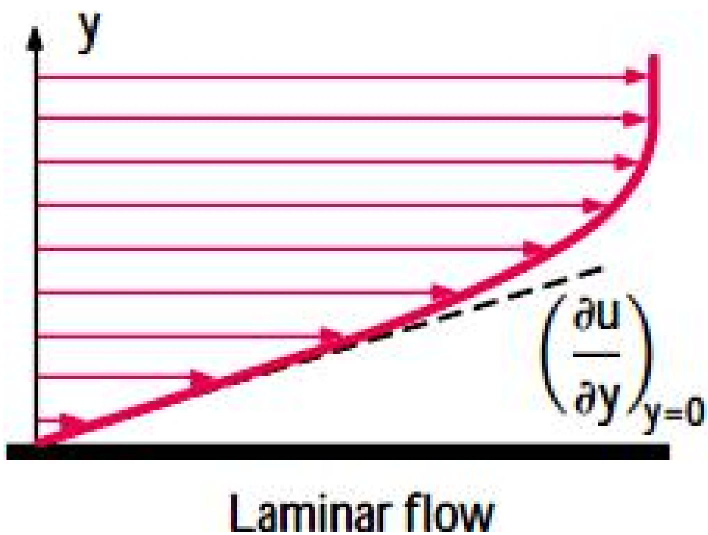
$$\tau_{total} = (\mu + \mu_t)\frac{\partial \bar{u}}{\partial y} = \rho(\nu + \nu_t)\frac{\partial \bar{u}}{\partial y}$$

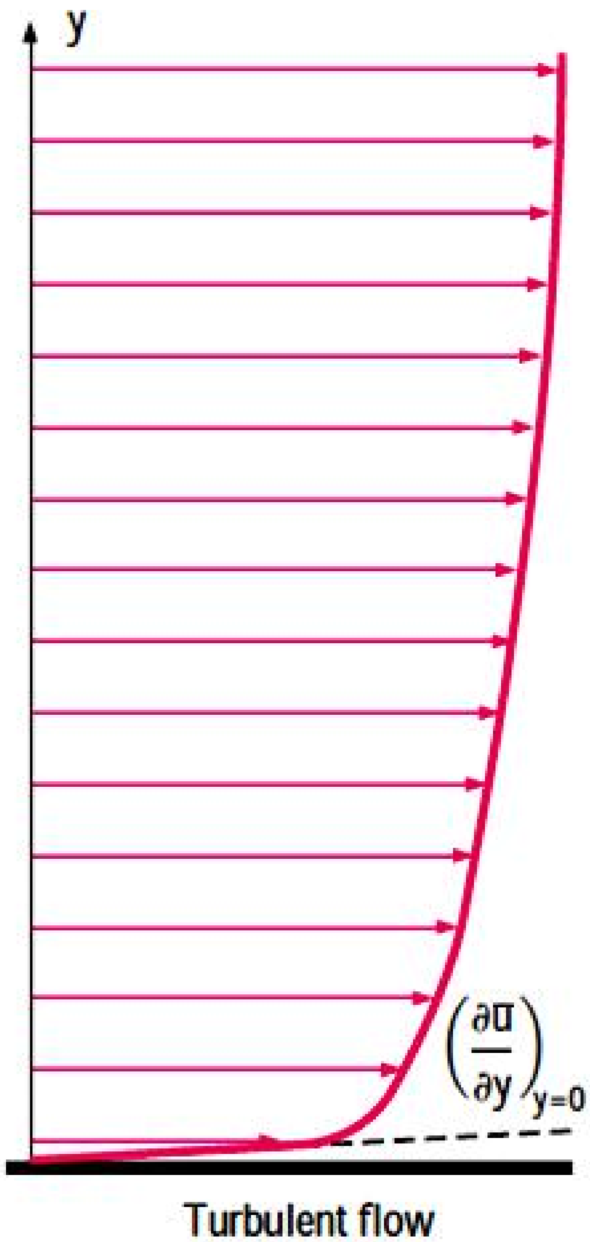
where μ_t is the **eddy viscosity** or **turbulent viscosity,** which accounts for momentum transport by turbulent eddies.

$\nu_t = \mu_t/\rho$ is the **kinematic eddy viscosity** or **kinematic turbulent viscosity.**

Turbulent Flow in Pipes

The velocity gradients at the wall, and thus the wall shear stress, are much larger for turbulent flow than they are for laminar flow.



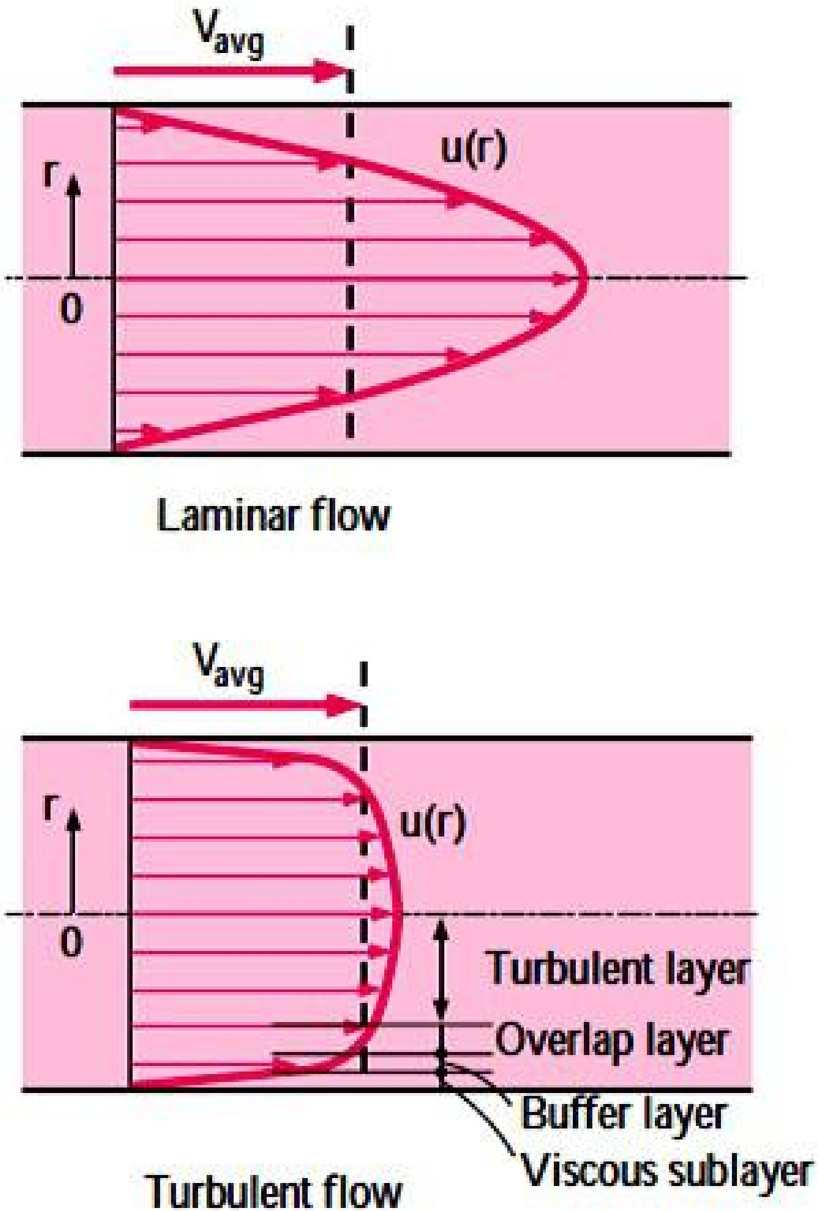


Turbulent Flow in Pipes

Turbulent Velocity Profile

The velocity profile is parabolic in laminar flow but is much **fuller** in turbulent flow, with a sharp drop near the pipe wall.

Turbulent flow along a wall can be considered to consist of four regions, characterized by the distance from the wall. The very thin layer next to the wall where viscous effects are dominant is the **viscous (**or **laminar** or **linear** or **wall)** sublayer.



Turbulent Flow in Pipes

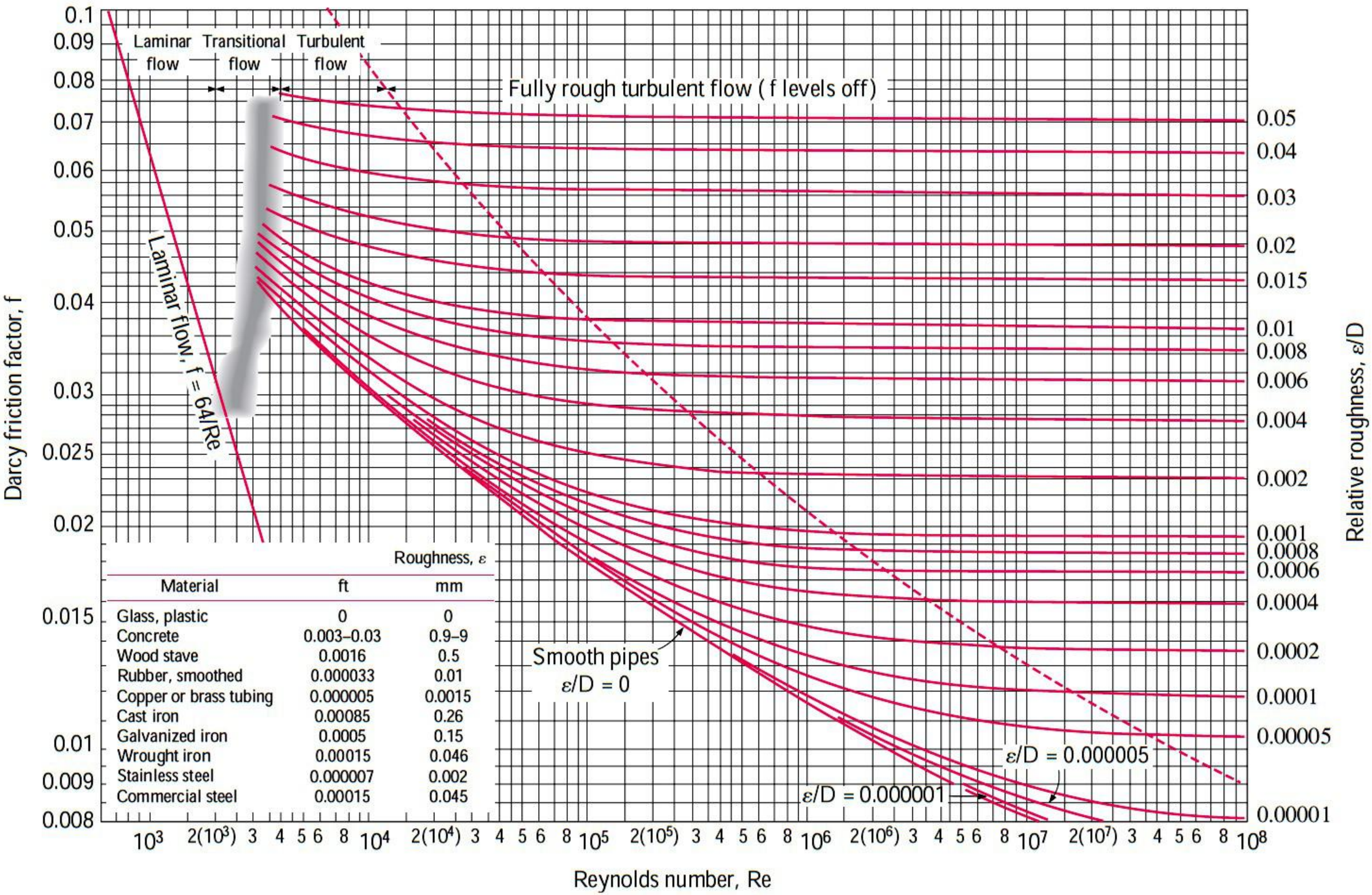
The velocity profile in this layer is very nearly linear, and the flow is streamlined.

Next to the viscous sublayer is the **buffer layer,** in which turbulent effects are becoming significant, but the flow is still dominated by viscous effects.

Above the buffer layer is the **overlap (or transition) layer,** also called th**e inertial sublayer,** in which the turbulent effects are much more significant, but still not dominant.

Above that is the **outer (or turbulent) layer in the remaining part of the flow in** which turbulent effects dominate over molecular diffusion (viscous) effects.

Flow characteristics are quite different in different regions, and thus **it is difficult to come up with an analytic relation for the velocity profile for the entire flow** as we did for laminar flow.



Material	Roughness, ε ft	Roughness, ε mm
Glass, plastic	0	0
Concrete	0.003–0.03	0.9–9
Wood stave	0.0016	0.5
Rubber, smoothed	0.000033	0.01
Copper or brass tubing	0.000005	0.0015
Cast iron	0.00085	0.26
Galvanized iron	0.0005	0.15
Wrought iron	0.00015	0.046
Stainless steel	0.000007	0.002
Commercial steel	0.00015	0.045

The Moody Chart

The Moody chart for friction factor for fully developed flow in circular pipes for use in the head loss relation

$$h_L = f \frac{L}{D} \frac{V^2}{2g}.$$

Friction factors in the turbulent flow are evaluated from the Colebrook equation

$$\frac{1}{\sqrt{f}} = -2 \log_{10}\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{Re\ \sqrt{f}}\right)$$

The Moody Chart

Equivalent roughness values for new commercial pipes*

Material	Roughness, ε ft	Roughness, ε mm
Glass, plastic	0 (smooth)	
Concrete	0.003–0.03	0.9–9
Wood stave	0.0016	0.5
Rubber, smoothed	0.000033	0.01
Copper or brass tubing	0.000005	0.0015
Cast iron	0.00085	0.26
Galvanized iron	0.0005	0.15
Wrought iron	0.00015	0.046
Stainless steel	0.000007	0.002
Commercial steel	0.00015	0.045

* The uncertainty in these values can be as much as ±60 percent.

Relative Roughness, ε/D	Friction Factor, f
0.0*	0.0119
0.00001	0.0119
0.0001	0.0134
0.0005	0.0172
0.001	0.0199
0.005	0.0305
0.01	0.0380
0.05	0.0716

* Smooth surface. All values are for Re = 10^6 and are calculated from the Colebrook equation.

Turbulent Flow in Pipes

The Colebrook equation is implicit in f, and thus the determination of the friction factor requires some iteration unless an equation solver is used.

An approximate explicit relation for f was given by **S. E. Haaland** in 1983 as

$$\frac{1}{\sqrt{f}} \cong -1.8 \log\left[\frac{6.9}{Re} + \left(\frac{\varepsilon/D}{3.7}\right)^{1.11}\right]$$

The results obtained from this relation are within 2 percent of those obtained from the Colebrook equation.

Turbulent Flow in Pipes

Types of Fluid Flow Problems

In the design and analysis of piping systems that involve the use of the Moody chart (or the Colebrook equation), we usually encounter three types of problems (the fluid and the roughness of the pipe are assumed to be specified in all cases)

1. Determining the **pressure drop (or head loss)** when the pipe length and diameter are given for a specified flow rate (or velocity)
2. Determining the **flow rate** when the pipe length and diameter are given for a specified pressure drop (or head loss)
3. Determining the **pipe diameter** when the pipe length and flow rate are given for a specified pressure drop (or head loss)

Turbulent Flow in Pipes

Problems of the first type are straightforward and can be solved **directly by using the Moody chart**.

Problems of the second type and third type are commonly encountered in engineering design (in the selection of pipe diameter, for example, that minimizes the sum of the construction and pumping costs), but the use of the Moody chart with such problems **requires an iterative approach** unless an equation solver is used.

To avoid tedious iterations in head loss, flow rate, and diameter calculations, **Swamee and Jain** proposed the following explicit relations in 1976 that are accurate to within 2 percent of the Moody chart:

Turbulent Flow in Pipes

$$h_L = 1.07\frac{\dot{V}^2 L}{gD^5}\left\{\ln\left[\frac{\varepsilon}{3.7D} + 4.62\left(\frac{\nu D}{\dot{V}}\right)^{0.9}\right]\right\}^{-2} \qquad \begin{matrix} 10^{-6} < \varepsilon/D < 10^{-2} \\ 3000 < Re < 3\times 10^{8}\end{matrix}$$

$$\dot{V} = -0.965\left(\frac{gD^5 h_L}{L}\right)^{0.5}\ln\left[\frac{\varepsilon}{3.7D} + \left(\frac{3.17\nu^2 L}{gD^3 h_L}\right)^{0.5}\right] \qquad Re > 2000$$

$$D = 0.66\left[\varepsilon^{1.25}\left(\frac{L\dot{V}^2}{gh_L}\right)^{4.75} + \nu\dot{V}^{9.4}\left(\frac{L}{gh_L}\right)^{5.2}\right]^{0.04} \qquad \begin{matrix} 10^{-6} < \varepsilon/D < 10^{-2} \\ 5000 < Re < 3\times 10^{8}\end{matrix}$$

THANK YOU