

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 0 - Visual fluids examples

Fluid dynamics is a unique subject because it's very visual. The book "An Album of Fluid Motion" by Milton van Dyke (on reserve at Wendt) is a collection of fascinating images from fluids experiments. You really can't say you understand fluids if you only think of it in terms of equations.

Figure 1 is a top view of a triangular wing immersed in a flow of water (moving from left to right in the image). Colored fluid, appearing as white, is introduced near the leading edge. The wing

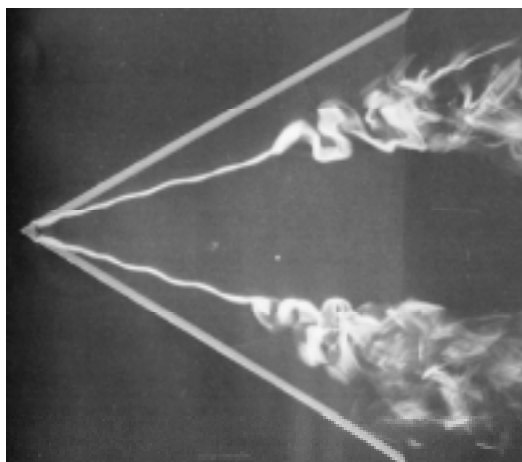


Figure 1: Turbulent transition in flow over a wing.

is inclined at a 20° angle of attack. Initially the fluid pattern is very smooth, then the filaments of colored fluid make a very abrupt transition to turbulence. The abruptness of the turbulent transition is interesting for many reasons, not least of which is that it's not really predicted by the equations of fluid flow.

Another interesting property of fluid flows is that the organization of it is very persistent. The upper left photo in Fig. 2 is of the wake of a circular cylinder in a water flow. The cylinder is at the left edge of the photo. The mean flow is very slow – about one cylinder diameter per second. The Reynolds number is 105. The alternating pattern of eddies (vortices) is called the 'Kàrmàn vortex street.' Intuitively, it kind of makes sense that a slow, laminar flow would be very organized. The upper right photo in Fig. 2, showing the wake of a plate at a 45° angle of attack, is taken at a flow that is, relatively, about 40 times faster (Reynolds number 4300). The flow in this case is turbulent, but even so, the alternating pattern of eddies is visible above the randomness. To drive home this point further, the lower photo in the figure is of the wake of a tanker inclined at roughly 45° to the mean current. The pattern assumed by the oil slick is amazingly similar to that in the upper right photo, even though the Reynolds number of the ship wake is on the order of 10^7 .

The sheer range of length scales that are interesting in fluids problems is pretty remarkable too. Active research ranges from flow in microchannels (blood flow in capillaries, for example) all the way up to cosmic systems like nebulae. The upper left photo in Fig. 3 shows the Kelvin-Helmholtz instability, which arises in the interface between flow streams at different velocities. The photo shows a rectangular tube (the 18 inch ruler in the image shows the scale), with pure water moving left-to-right on top and colored salt water moving right-to-left below. The upper image on the right is of clouds near Denver. On the leeward (sheltered) side of mountain ranges, you will often have layers of high winds above relatively slow air. When clouds sit near the interfaces, these Kelvin-Helmholtz structures result. Needless to say, the cloud structures are much bigger than those seen in the lab, but the math is the same. (The cloud image is taken from "Hydrodynamic Stability" by Drazin and Reid, also on reserve.)

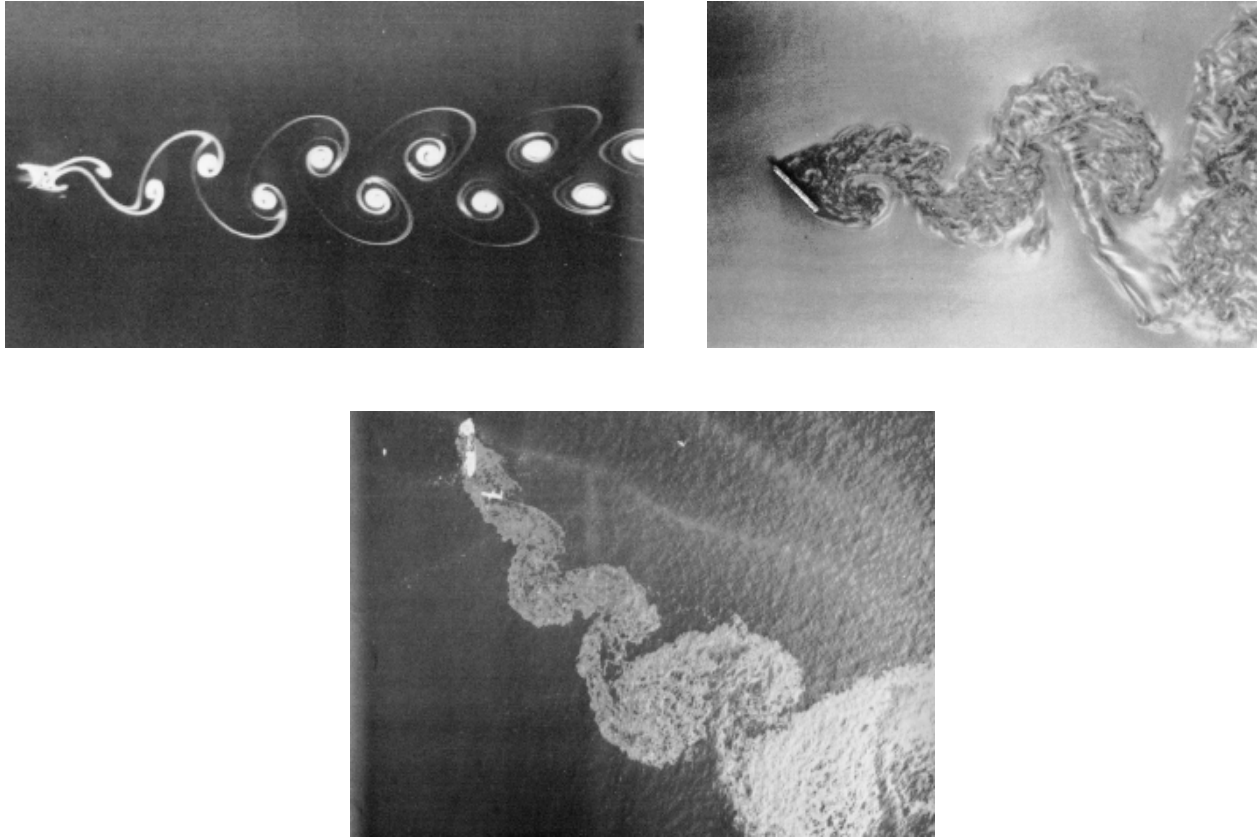


Figure 2: Persistence of flow organization, for Reynolds numbers spanning five orders of magnitude.

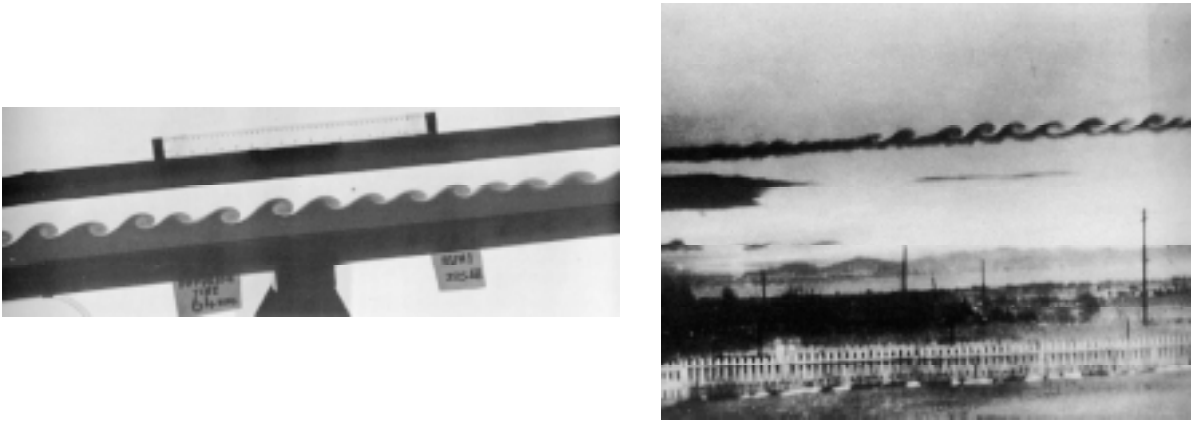


Figure 3: Fluid problems span a vast range of length scales.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 1 - Math fundamentals

The subject of fluid dynamics brings to mind pipes, pumps, wind tunnels, fans and any number of other engineering devices. However, to study fluid dynamics effectively requires some basic mathematical tools. (Interestingly, while fluid dynamics is thought to be very much the domain of engineers, much work in fluids, particularly in Europe, is performed in applied math, math, applied physics and physics departments.) In this class it will be assumed that you are familiar with the basic techniques of differentiation and integration, partial derivatives, differential equations and vector calculus. You may not remember everything from your math courses but you should at least know where to find things (tables of integrals, for example). What is important is your mathematical intuition.

The following expression should look vaguely familiar:

$$\lim_{\Delta x \rightarrow 0} \frac{f(x_0 + \Delta x) - f(x_0 - \Delta x)}{2\Delta x} = ?$$

If a function $f(x)$ is differentiable at a point x_0 , then this gives the value of the derivative, $f'(x_0)$, at that point, that is

$$\lim_{\Delta x \rightarrow 0} \frac{f(x_0 + \Delta x) - f(x_0 - \Delta x)}{2\Delta x} = f'(x_0) \quad \text{if } f \text{ is differentiable at } x_0. \quad (1)$$

This expression gives the value of $f'(x_0)$ if f is differentiable, but just because the limit in (1) exists doesn't mean that f is differentiable at x_0 (what's a counterexample?). It turns out, if the one-sided limits are equal –

$$\lim_{\Delta x \rightarrow 0} \frac{f(x_0 + \Delta x) - f(x_0)}{\Delta x} = \lim_{\Delta x \rightarrow 0} \frac{f(x_0) - f(x_0 - \Delta x)}{\Delta x} \quad (\text{with } \Delta x > 0) \quad (2)$$

then f is differentiable at x_0 , and $f'(x_0)$ is equal to the value of the one-sided limits of (2), which is then equal to the two-sided limit of (1).

Thinking of derivatives in terms of differences (hopefully) seems completely trivial, but understanding (1) and (2) implicitly is key to subjects like fluids that rely heavily on differential equations (and if it were easy, it wouldn't have taken Isaac Newton to come up with calculus). This is especially true since data (whether experimental or computational) is digital, and discrete math is used extensively.

1 Vector analysis

Vector analysis is vital to fluid mechanics because the most fundamental descriptive quantity in fluid flow, the velocity, is a vector. We'll start with the three-dimensional Cartesian coordinate system with unit vectors \hat{e}_x , \hat{e}_y and \hat{e}_z . Consider the vectors $\mathbf{a} = a_x\hat{e}_x + a_y\hat{e}_y + a_z\hat{e}_z$ and $\mathbf{b} = b_x\hat{e}_x + b_y\hat{e}_y + b_z\hat{e}_z$. The dot product (or scalar product) of \mathbf{a} and \mathbf{b} is

$$\mathbf{a} \cdot \mathbf{b} = |\mathbf{a}| |\mathbf{b}| \cos \theta = a_x b_x + a_y b_y + a_z b_z \quad (3)$$

where $|\cdot|$ denotes the length, or magnitude, of a vector, and θ is the angle between the vectors.

The cross product (or vector product) of \mathbf{a} and \mathbf{b} , $\mathbf{a} \times \mathbf{b}$, is more complicated. Its magnitude is given by

$$|\mathbf{a} \times \mathbf{b}| = |\mathbf{a}| |\mathbf{b}| \sin \theta$$

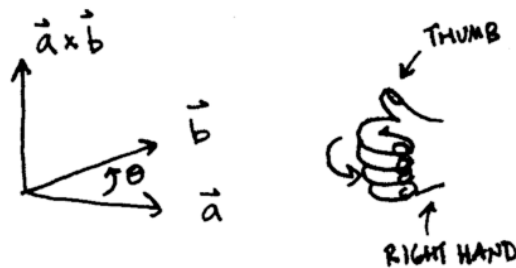


Figure 1: The direction of the cross product $\mathbf{a} \times \mathbf{b}$.

where θ is the (smallest) angle between \mathbf{a} and \mathbf{b} (i.e. $\theta \leq 180^\circ$). The direction of $\mathbf{a} \times \mathbf{b}$ is perpendicular to the plane of \mathbf{a} and \mathbf{b} , in the right-hand sense of rotation from \mathbf{a} to \mathbf{b} through the angle θ (Fig. 1). Because of this direction definition, the cross product anti-commutes, i.e. $\mathbf{a} \times \mathbf{b} = -\mathbf{b} \times \mathbf{a}$.

It turns out that the most convenient way to express the cross product is as a determinant –

$$\mathbf{a} \times \mathbf{b} = \begin{vmatrix} \hat{e}_x & \hat{e}_y & \hat{e}_z \\ a_x & a_y & a_z \\ b_x & b_y & b_z \end{vmatrix} = (a_y b_z - a_z b_y)\hat{e}_x + (a_z b_x - a_x b_z)\hat{e}_y + (a_x b_y - a_y b_x)\hat{e}_z \quad (4)$$

Easily seen through (4) are the identities $\hat{e}_x \times \hat{e}_y = \hat{e}_z$, $\hat{e}_y \times \hat{e}_z = \hat{e}_x$, and $\hat{e}_z \times \hat{e}_x = \hat{e}_y$.

Of particular interest is the operator ∇ (variously called the gradient operator, del operator, or grad operator). We can write ∇ as

$$\nabla = \hat{e}_x \frac{\partial}{\partial x} + \hat{e}_y \frac{\partial}{\partial y} + \hat{e}_z \frac{\partial}{\partial z} \quad (5)$$

∇ has no meaning unless it's operating on something. The familiar operations are the gradient, the divergence, the curl, and the Laplacian.

1.1 The gradient and directional derivative

When ∇ operates on a scalar quantity, $\phi(x, y, z)$, the result is called the *gradient* of ϕ , and is written

$$\nabla \phi = \hat{e}_x \frac{\partial \phi}{\partial x} + \hat{e}_y \frac{\partial \phi}{\partial y} + \hat{e}_z \frac{\partial \phi}{\partial z} \quad (6)$$

To interpret this, consider a unit vector $\hat{\mathbf{s}} = s_x \hat{e}_x + s_y \hat{e}_y + s_z \hat{e}_z$, and let s be the distance variable along this vector. Without loss of generality, we will assume that $s = 0$ at the origin, $\mathbf{x} = 0$. Then, the coordinates of any point on the s -axis are

$$\begin{aligned} x &= s_x s \\ y &= s_y s \\ z &= s_z s \end{aligned} \quad (7)$$

Suppose we want to know the rate of change of ϕ in the s direction, at the origin. This $d\phi/ds$ is also known as the directional derivative. By the chain rule, this can be written (note the selective use of partial derivatives)

$$\begin{aligned} \frac{d\phi}{ds} &= \frac{\partial \phi}{\partial x} \frac{dx}{ds} + \frac{\partial \phi}{\partial y} \frac{dy}{ds} + \frac{\partial \phi}{\partial z} \frac{dz}{ds} \\ &= \frac{\partial \phi}{\partial x} s_x + \frac{\partial \phi}{\partial y} s_y + \frac{\partial \phi}{\partial z} s_z \end{aligned} \quad (8)$$

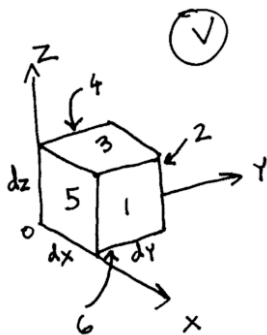


Figure 2: Sample volume for illustrating the divergence.

where we have used (7). By inspection, we have

$$\frac{d\phi}{ds} = \nabla\phi \cdot \hat{\mathbf{s}} \quad (9)$$

that is, *the rate of change of a scalar function ϕ in an arbitrary direction is equal to the scalar product of the gradient $\nabla\phi$ with the unit vector, $\hat{\mathbf{s}}$, in that direction.* Using (3), we can also write (since $|\hat{\mathbf{s}}|=1$)

$$\frac{d\phi}{ds} = |\nabla\phi| \cos\theta \quad (10)$$

where θ is the angle between $\nabla\phi$ and $\hat{\mathbf{s}}$. So we can also say that $d\phi/ds$ is the projection of $\nabla\phi$ onto the direction \mathbf{s} . Finally, observe from (10) that $\nabla\phi$ is perpendicular to lines of constant ϕ .

1.2 The divergence and divergence theorem

Now let's consider what happens when we apply ∇ to vector quantities. Consider a vector field $\mathbf{f} = f_x\hat{\mathbf{e}}_x + f_y\hat{\mathbf{e}}_y + f_z\hat{\mathbf{e}}_z$. (We call this a vector 'field' because it's defined over a volume in space, not just at a single point.) The dot product of ∇ with \mathbf{f} is called the *divergence*, and is given by

$$\nabla \cdot \mathbf{f} = \frac{\partial f_x}{\partial x} + \frac{\partial f_y}{\partial y} + \frac{\partial f_z}{\partial z} \quad (11)$$

To understand the name 'divergence', let the three-dimensional volume V be a cube with infinitesimal side lengths $dx = dy = dz$, as in Fig. 2. We can write the volume of V as $dx dy dz = dV$. Assume that on each face of the cube, \mathbf{f} is constant. Now consider face 1 in the figure. The component of \mathbf{f} that points out of the cube on face 1 is f_x . Because \mathbf{f} is constant on the face, we can write this as $f_x(x = dx)$. On face 4, the component of \mathbf{f} that points out of the cube is $-f_x(x = 0)$. The area of each of these two faces is $dy \cdot dz$, so the net volume flux out of the cube through faces 1 and 4 can be written

$$\text{Flux out of faces 1 and 4} = [f_x(x = dx) - f_x(x = 0)] dy dz \quad (12)$$

We can go through similar arguments for the remaining faces, and we get

$$\begin{aligned} \text{Flux out of faces 2 and 5} &= [f_y(y = dy) - f_y(y = 0)] dx dz \\ \text{Flux out of faces 3 and 6} &= [f_z(z = dz) - f_z(z = 0)] dx dy \end{aligned} \quad (13)$$

Now notice that we can rewrite (12) as

$$\begin{aligned}
 [f_x(x = dx) - f_x(x = 0)] dy dz &= \frac{f_x(x = dx) - f_x(x = 0)}{dx} dx dy dz \\
 &= \frac{f_x(x = dx) - f_x(x = 0)}{dx} dV \\
 &\approx \frac{\partial f_x}{\partial x} dV
 \end{aligned} \tag{14}$$

where the last part is true in the limit of dx approaching zero. Similarly, the flux out of faces 2 and 5 can be written $(\partial f_y / \partial y) dV$, and out of faces 3 and 6 can be written $(\partial f_z / \partial z) dV$. Referring back to (11), given a vector \mathbf{f} , *the net volume flux out of the volume dV equals the product of $\nabla \cdot \mathbf{f}$ and dV* . Thus $\nabla \cdot \mathbf{f}$ is a measure of the spread, or divergence, of the vector field \mathbf{f} .

This somewhat non-rigorous analysis can be generalized as the **divergence theorem**. Given an arbitrary volume V , enclosed by the surface S , with outward unit normal vector \mathbf{n} at all points on S , the divergence theorem states

$$\int_V \nabla \cdot \mathbf{f} dV = \int_S \mathbf{f} \cdot \mathbf{n} dS. \tag{15}$$

This will be very useful when we get to the equations for fluid flow.

1.3 The curl and Stokes' theorem

The cross product of ∇ with the vector \mathbf{f} is called the *curl*, and is given by

$$\nabla \times \mathbf{f} = \begin{vmatrix} \hat{e}_x & \hat{e}_y & \hat{e}_z \\ \frac{\partial}{\partial x} & \frac{\partial}{\partial y} & \frac{\partial}{\partial z} \\ f_x & f_y & f_z \end{vmatrix} = \left(\frac{\partial f_z}{\partial y} - \frac{\partial f_y}{\partial z} \right) \hat{e}_x + \left(\frac{\partial f_x}{\partial z} - \frac{\partial f_z}{\partial x} \right) \hat{e}_y + \left(\frac{\partial f_y}{\partial x} - \frac{\partial f_x}{\partial y} \right) \hat{e}_z \tag{16}$$

To see where the term ‘curl’ comes from, look at Fig. 2 again, but this time only consider face 6. Call this the surface S , with area $dx dy = dS$. We’ll define a direction of travel around the border of S in the counter-clockwise direction. Assume also that the vector field \mathbf{f} is constant on each edge of S . Starting at the the origin, we first travel along the edge defined by $y = 0$. The component of \mathbf{f} parallel to the direction of travel on this edge is $f_x(y = 0)$. We can then define a contour integral (a sort of net travel) as $f_x(y = 0) dx$. The next edge is the one defined by $x = dx$, along which the net travel $f_y(x = dx) dy$. Going all the way back around to the origin, we end up with (taking care with the signs)

$$\begin{aligned}
 \text{Net travel} &= f_x(y = 0) dx + f_y(x = dx) dy - f_x(y = dy) dx - f_y(x = 0) dy \\
 &= [f_y(x = dx) - f_y(x = 0)] dy - [f_x(y = dy) - f_x(y = 0)] dx \\
 &= \frac{f_y(x = dx) - f_y(x = 0)}{dx} dS - \frac{f_x(y = dy) - f_x(y = 0)}{dy} dS \\
 &\approx \left(\frac{\partial f_y}{\partial x} - \frac{\partial f_x}{\partial y} \right) dS
 \end{aligned} \tag{17}$$

Again, this last expression is true for dx and dy approaching zero. Comparing with (16), (17) is just the z -component of $\nabla \times \mathbf{f}$. By the direction convention for integration around a closed contour, \hat{e}_z is the normal vector for face 6 (our surface S) in Fig. 2 for the counter-clockwise integration direction. Thus, *the integral of \mathbf{f} around the contour enclosing S equals the component of $\nabla \times \mathbf{f}$ in the direction normal to S multiplied by dS* . In the context of the integration around the closed contour, the use of the term ‘curl’ is obvious.



Figure 3: Direction convention for integration around a closed contour, C . The vector $\hat{\mathbf{n}}$ is the unit normal to the surface.

Stokes' theorem generalizes this. Let S be a two-dimensional surface enclosed by the curve C . Then

$$\int_S (\nabla \times \mathbf{f}) \cdot \mathbf{n} dS = \int_C \mathbf{f} \cdot d\mathbf{x} \quad (18)$$

where the direction of the line integral relates to the direction of the normal vector \mathbf{n} in the right-hand sense (Fig. 3).

1.4 The Laplacian

If we take the divergence of a gradient, we get the Laplacian, which is defined (ϕ is a scalar)

$$\nabla^2 \phi = \nabla \cdot (\nabla \phi) = \frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} \quad (19)$$

applying (6) and (11). The Laplacian is interesting in relation to quantities that undergo gradient diffusion. An example is heat, which diffuses proportionally to the gradient in temperature. If we let ϕ be the temperature and κ be the thermal diffusivity, then the temperature flux vector can be written (note the minus sign)

$$\text{Temperature flux vector} = -\kappa \nabla \phi \quad (20)$$

The Laplacian can thus be interpreted by substituting $-\nabla \phi$ for \mathbf{f} in the discussion of the divergence in Sec. 1.2. That is, the Laplacian describes the net flux of the scalar quantity *into* a volume. Consider a uniform temperature field with a sharp positive spike somewhere in it. The spike will have a strongly negative Laplacian value, which means that heat will flow out from the region of the spike, which makes sense, since diffusivity tends to smooth out sharp gradients. Another scalar quantity that undergoes gradient diffusion is species concentration – a *vector* quantity that does is momentum, but we will need more math tools to consider the diffusion of vector quantities.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 2 - More math, plus some basic physics

In the first lecture we went over some basic math concepts, in particular the operator ∇ . We'll finish up with basic math by going over tensors and index notation, then talk about some basic physical concepts.

1 Tensors and index notation

In the last lecture we considered the dot product, where two vectors result in a scalar, and the cross product, where the two vectors yield a third vector. There is another way to multiply vectors together that gives rise not to a scalar or a vector, but to a *tensor*. To deal with those it's most convenient to use index notation (also called tensor notation).

1.1 Index notation

Scalars and vectors are actually specific cases of tensors. A scalar is a tensor of order (or rank) zero, and a vector is a tensor of order one. (Generally, however, if we say that something is a tensor without specifying its order, we will mean that it is a tensor of order two.) The order of a tensor tells you the number of indices necessary to describe it. In n -dimensional space, a tensor of order m has n^m components.

We can repeat some of the results of the last lecture using index notation. Consider a vector \mathbf{f} defined in three-dimensional space. Let the three orthogonal unit vectors be \hat{e}_1 , \hat{e}_2 , and \hat{e}_3 . (We're not using x , y and z because the coordinate system is not necessarily Cartesian, or it could be rotated from x , y and z , etc.) Then we can write \mathbf{f} as

$$\mathbf{f} = f_1\hat{e}_1 + f_2\hat{e}_2 + f_3\hat{e}_3 = f_i\hat{e}_i \quad (1)$$

This expression illustrates two key aspects of index notation:

- An index (in this case i) takes on values corresponding to the number of dimensions in the space being considered.
- If an index is repeated in a term, then that term is summed over that index (this is the summation convention, which physicists call the Einstein summation convention).

The dot product and cross product of two vectors can be expressed conveniently in index notation, with the aid of two new operators. The dot product of two vectors \mathbf{a} and \mathbf{b} is

$$\mathbf{a} \cdot \mathbf{b} = a_i b_j \delta_{ij}, \text{ where } \delta_{ij} = \begin{cases} 1 & \text{if } i = j \\ 0 & \text{if } i \neq j. \end{cases} \quad (2)$$

This δ_{ij} is called the Kronecker delta. (Of course, we could also have written $\mathbf{a} \cdot \mathbf{b} = a_i b_i$ using the summation convention.) The cross product of \mathbf{a} and \mathbf{b} is

$$\mathbf{a} \times \mathbf{b} = \epsilon_{ijk} a_j b_k \hat{e}_i, \text{ where } \epsilon_{ijk} = \begin{cases} 1 & \text{if } ijk \text{ is cyclic, i.e. } 123, 231, \text{ or } 312 \\ -1 & \text{if } ijk \text{ is anti-cyclic, i.e. } 321, 213, 132 \\ 0 & \text{if any of the two indices are identical.} \end{cases} \quad (3)$$

The ϵ_{ijk} is called the permutation tensor or permutation operator.

1.2 Tensors

Consider two vectors $\mathbf{a} = a_1\hat{e}_1 + a_2\hat{e}_2 + a_3\hat{e}_3 = a_i\hat{e}_i$ and $\mathbf{b} = b_1\hat{e}_1 + b_2\hat{e}_2 + b_3\hat{e}_3 = b_i\hat{e}_i$. If we multiply them together, *not* as a dot product or cross product, but just \mathbf{ab} , we get $\mathbf{ab} = \mathbf{T}$, or

$$\mathbf{ab} = a_i b_j = \begin{pmatrix} a_1 b_1 & a_1 b_2 & a_1 b_3 \\ a_2 b_1 & a_2 b_2 & a_2 b_3 \\ a_3 b_1 & a_3 b_2 & a_3 b_3 \end{pmatrix} = T_{ij} = \mathbf{T} \quad (4)$$

where, by convention, the first index, here i , represents the row of the tensor and the second index, j , represents the column. Also, a quick notational point. The order of a tensor is sometimes identified by underlining. So a vector \mathbf{a} is often written \underline{a} , and a tensor \mathbf{T} is often written $\underline{\underline{T}}$. The underlining convention is common with written work; in printed work, vectors and tensors are usually just represented in boldface (some authors use lowercase bold for vectors and uppercase bold for tensors, but this isn't universal).

Because the first index in a tensor represents the row, and a vector is a tensor of order one, the vector \mathbf{a} can be written

$$\mathbf{a} = a_i \hat{e}_i = \begin{pmatrix} a_1 \\ a_2 \\ a_3 \end{pmatrix}. \quad (5)$$

The matrix representation of \mathbf{a} is not written with the unit vectors \hat{e}_i because in tensor form, the unit vectors are assumed to go with the components i (this can sometimes be confusing).

The *transpose* of a tensor is a tensor with its rows and columns switched. For a second-order tensor \mathbf{T} , this means the transpose \mathbf{T}^T is given by

$$\text{for } \mathbf{T} = T_{ij} = \begin{pmatrix} T_{11} & T_{12} & T_{13} \\ T_{21} & T_{22} & T_{23} \\ T_{31} & T_{32} & T_{33} \end{pmatrix}, \text{ the transpose is } \mathbf{T}^T = T_{ji} = \begin{pmatrix} T_{11} & T_{21} & T_{31} \\ T_{12} & T_{22} & T_{32} \\ T_{13} & T_{23} & T_{33} \end{pmatrix}. \quad (6)$$

In the expression for the transpose, when we write $\mathbf{T}^T = T_{ji}$, the first index still corresponds to the row even though we use j instead of i . A tensor \mathbf{T} is called *symmetric* if $\mathbf{T} = \mathbf{T}^T$, i.e. if $T_{ij} = T_{ji}$. A tensor \mathbf{T} is *anti-symmetric* if $\mathbf{T} = -\mathbf{T}^T$, or $T_{ij} = -T_{ji}$.

For a vector \mathbf{a} , taking the transpose just means

$$\text{for } \mathbf{a} = \begin{pmatrix} a_1 \\ a_2 \\ a_3 \end{pmatrix}, \text{ the transpose is } \mathbf{a}^T = (a_1 \ a_2 \ a_3). \quad (7)$$

The standard rules of matrix multiplication can be applied to tensors. Matrix multiplication corresponds to the dot product. It is possible to take the dot product of a tensor \mathbf{T} and a vector \mathbf{a} –

$$\mathbf{T} \cdot \mathbf{a} = T_{ij} a_j = \begin{pmatrix} T_{11} & T_{12} & T_{13} \\ T_{21} & T_{22} & T_{23} \\ T_{31} & T_{32} & T_{33} \end{pmatrix} \begin{pmatrix} a_1 \\ a_2 \\ a_3 \end{pmatrix} = \begin{pmatrix} T_{11}a_1 + T_{12}a_2 + T_{13}a_3 \\ T_{21}a_1 + T_{22}a_2 + T_{23}a_3 \\ T_{31}a_1 + T_{32}a_2 + T_{33}a_3 \end{pmatrix} \quad (8)$$

(Notice the application of the summation convention.) The dot product in this case *doesn't* yield a scalar, but instead a vector. The dot product essentially gives a result one order reduced from the highest order tensor in the product. This leads to the other form of multiplication we'll be interested in, the double-dot product (also known as the scalar product for tensors). This takes two second-order tensors and yields a scalar, as

$$\mathbf{T} : \mathbf{T} = T_{ij} T_{ij}. \quad (9)$$

1.3 Velocity gradient tensor

The tensor we will be most interested in is the velocity gradient tensor, $\nabla \mathbf{u}$. For Cartesian coordinates, with $\mathbf{u} = u\hat{e}_x + v\hat{e}_y + w\hat{e}_z$, $\nabla \mathbf{u}$ is defined as

$$\nabla \mathbf{u} = \frac{\partial u_j}{\partial x_i} = \begin{pmatrix} \frac{\partial u}{\partial x} & \frac{\partial v}{\partial x} & \frac{\partial w}{\partial x} \\ \frac{\partial u}{\partial y} & \frac{\partial v}{\partial y} & \frac{\partial w}{\partial y} \\ \frac{\partial u}{\partial z} & \frac{\partial v}{\partial z} & \frac{\partial w}{\partial z} \end{pmatrix} \quad (10)$$

The Laplacian of the velocity, $\nabla^2 \mathbf{u}$, will also appear often in our discussions. This is found by

$$\begin{aligned} \nabla^2 \mathbf{u} = \nabla \cdot (\nabla \mathbf{u}) &= \begin{pmatrix} \frac{\partial}{\partial x} & \frac{\partial}{\partial y} & \frac{\partial}{\partial z} \end{pmatrix} \begin{pmatrix} \frac{\partial u}{\partial x} & \frac{\partial v}{\partial x} & \frac{\partial w}{\partial x} \\ \frac{\partial u}{\partial y} & \frac{\partial v}{\partial y} & \frac{\partial w}{\partial y} \\ \frac{\partial u}{\partial z} & \frac{\partial v}{\partial z} & \frac{\partial w}{\partial z} \end{pmatrix} \\ &= \begin{pmatrix} \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \\ \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \\ \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \end{pmatrix} \\ &= \begin{pmatrix} \nabla^2 u \\ \nabla^2 v \\ \nabla^2 w \end{pmatrix} \end{aligned} \quad (11)$$

so taking the Laplacian of the velocity vector is equivalent to taking the Laplacian of each of the components.

(We'll talk about the physical meaning of $\nabla \mathbf{u}$ in detail later.)

2 Physical concepts

Let's now discuss some basic physical ideas. One of the goals of this class is to develop the ability to think through problems intuitively before having to bring math tools to bear. Suppose I have two airplanes. They are identical except one of them has winglets on the wingtips, like you see on some airliners (747's, A340's etc.).

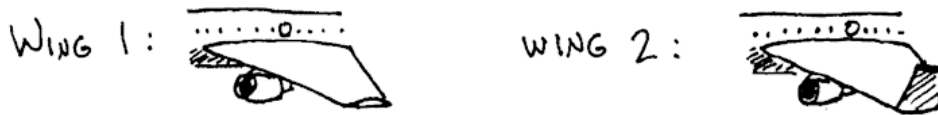


Figure 1: Two wings: wing 1, no winglet, wing 2, winglet.

The first wing shows a tip vortex, while the second doesn't.

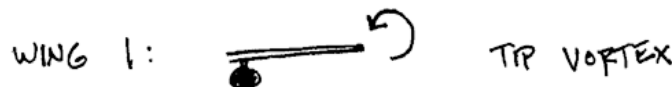


Figure 2: Tip vortex on wing 1.

Why does this tip vortex arise? First, we know that wings are designed to provide elevated pressures on the lower surface, and reduced pressures on the upper surface. However, if we pick a point just outside of the wingtip, the pressure has to be the same from above and below, because

the pressure can't have a discontinuity. As a result, the pressure on the underside of the wing is gradually reduced as we approach the tip, and the pressure above the wing is gradually increased as we approach the tip, so the pressures become equal outside the tip. Because fluids move in the

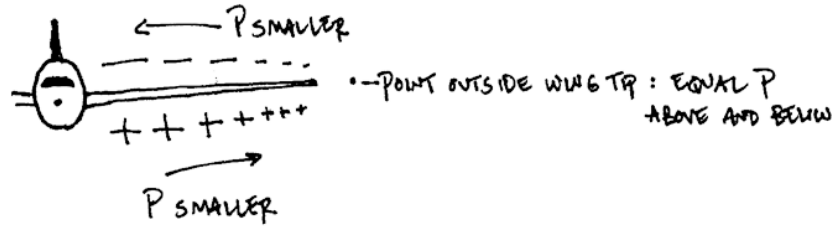


Figure 3: Pressure distribution around a wing.

direction of decreasing pressure, the airflow on the lower surface of the wing moves toward the tip, and the airflow on the lower surface moves toward the plane's fuselage. Put together, the result is a tip vortex with the sense of rotation shown in Fig. 2. (The winglet in wing 2 is designed to prevent the formation of this vortex.) Basically, two principles (and no math) were necessary to show this - the idea that pressure fields are continuous, and that flows are driven in the direction of decreasing pressure.

Why, though, would a designer want to get rid of the tip vortex? Let's consider what effect the tip vortex has by imagining that we're stationary in space, and can somehow see the velocity of the air. A plane equipped with winglets passes, leaving the air essentially undisturbed. A plane without winglets passes, and in contrast we see the air spinning off the wingtips. The subsequent reasoning goes like this:

- The flow with the vortical structures has more kinetic energy than the undisturbed flow.
- The kinetic energy has to be provided by the plane's engines.
- The energy provided by the engines to the vortices is energy *not* provided to the plane's forward motion.
- The vortices thus correlate with an effective drag force on the plane.

We've thus shown that the tip vortices cause drag, and we needed to apply only the conservation of energy, again with no math. Unfortunately...not all problems can be solved with just physical intuition, but physical intuition is still very handy.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 3 - Basic concepts of ideal fluids / Euler's equations

Reading: Acheson, chapter 1.

In describing fluid flow it is common to speak of the velocity field, $\mathbf{u} = \mathbf{u}(\mathbf{x}, t)$, which gives the value of the velocity vector, \mathbf{u} , at all points \mathbf{x} and times t . (In Cartesian coordinates, $\mathbf{u} = u\hat{e}_x + v\hat{e}_y + w\hat{e}_z = (u, v, w)$ and $\mathbf{x} = x\hat{e}_x + y\hat{e}_y + z\hat{e}_z = (x, y, z)$.) The following basic notions will be used extensively.

- A *steady flow* is one where the velocity is a function of spatial position only, i.e. $\mathbf{u} = \mathbf{u}(\mathbf{x})$, and $\partial\mathbf{u}/\partial t = 0$.
- A *two-dimensional flow* is one in which the velocity is independent of one spatial direction, and there is no velocity component in that direction. In Cartesian coordinates, $\mathbf{u} = u(x, y, t)\hat{e}_x + v(x, y, t)\hat{e}_y$ is an example of a two-dimensional flow.
- A *streamline* is a curve that is parallel to the local velocity vector at all points. Suppose s is a variable that measures the distance traveled along a particular streamline. In Cartesian coordinates, a streamline would be expressed $x = x(s)$, $y = y(s)$, $z = z(s)$. At any point on the streamline, $dx/dy = u/v$, $dy/dz = v/w$, and $dx/dz = u/w$ (compare Eq. 1.5 in the book).
- The *material derivative* Df/Dt describes the rate of change of a quantity f in a reference frame following the fluid. It is defined

$$\begin{aligned}\frac{Df}{Dt} &= \frac{\partial f}{\partial t} + u\frac{\partial f}{\partial x} + v\frac{\partial f}{\partial y} + w\frac{\partial f}{\partial z} \\ &= \frac{\partial f}{\partial t} + (\mathbf{u} \cdot \nabla)f = \frac{\partial f}{\partial t} + \mathbf{u} \cdot (\nabla f).\end{aligned}\tag{1}$$

The velocity components u , v and w appear courtesy of the chain rule. This Df/Dt is also known as the substantial derivative or particle derivative. The process described by Df/Dt is known as *advection*, and involves the change in f due only to the motion of the fluid. Other factors can affect f and must be dealt with separately (examples to follow).

1 Ideal fluids

An *ideal fluid* is one for which the following conditions hold –

1. It is incompressible. In practice, this generally means that the density, ρ , is a constant throughout the fluid.
2. It is inviscid. Thus, the only force acting across any surface element dS in the fluid is the pressure.

Strictly speaking, the property of incompressibility is better ascribed to the flow than the fluid. For example, gases are obviously compressible, but under certain conditions (specifically, low Mach number) their flows can be treated as incompressible.

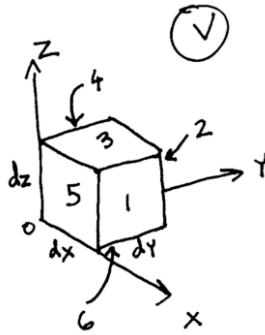


Figure 1: An infinitesimal cubic volume V .

1.1 Equations of motion for an ideal fluid

To get the equations of motion for an ideal fluid, we might reasonably expect to use the familiar conservation properties from physics: conservation of mass, momentum, and energy. Sure enough, the first equation of motion describes the conservation of mass. Let V be an arbitrary volume in the fluid, enclosed by the surface S . Because the density of the fluid is constant, the net volume flux across the surface S has to be zero. By the divergence theorem, this means –

$$\int_V \nabla \cdot \mathbf{u} dV = 0 \text{ for any arbitrary volume } V. \quad (2)$$

We can make this statement stronger. Suppose that $\nabla \cdot \mathbf{u}$ is positive (or negative) at some point \mathbf{x}_0 . Assuming that the $\nabla \cdot \mathbf{u}$ is continuous, then we can define a volume V around \mathbf{x}_0 such that $\nabla \cdot \mathbf{u}$ is positive (or negative) at all points in V . But then this violates (2). Thus, the conservation of mass leads to the first equation of motion for an ideal fluid, namely

$$\nabla \cdot \mathbf{u} = 0 \text{ everywhere in an ideal fluid.} \quad (3)$$

Now for the conservation of momentum. Again, we let V be some arbitrary volume enclosed by S . By the second property of ideal fluids listed above, the only forces acting across S are pressure forces. We are interested in the forces acting *on* the fluid in V ; we are not interested here in the forces applied *by* the fluid contained in V . The net force exerted on the fluid in V by the surrounding fluid is

$$- \int_S p \mathbf{n} dS \quad (4)$$

where the minus sign indicates that the force vector points into V in the direction of the normal vector to S . To cast this in a more convenient form, we'll again do an analysis similar to what we did to illustrate the divergence in Lecture 1. Figure 1 is the same infinitesimal cubic volume V , with volume $dV = dx dy dz$, that we used before. The pressure force applied by the surrounding fluid on face 1 is $-p(x = dx) dy dz \hat{e}_x$, and on face 4 is $p(x = 0) dy dz \hat{e}_x$. (As before, we have assumed that p is constant on each face.) The other faces can be treated analogously. We get

$$\begin{aligned} \text{Pressure force on faces 1 and 4} &= -[p(x = dx) - p(x = 0)] dy dz \hat{e}_x \\ \text{Pressure force on faces 2 and 5} &= -[p(y = dy) - p(y = 0)] dx dz \hat{e}_y \\ \text{Pressure force on faces 3 and 6} &= -[p(z = dz) - p(z = 0)] dx dy \hat{e}_z. \end{aligned} \quad (5)$$

Look at the force on faces 1 and 4. We can rewrite that component as

$$\begin{aligned} -[p(x = dx) - p(x = 0)] dy dz \hat{e}_x &= -\frac{p(x = dx) - p(x = 0)}{dx} dV \hat{e}_x \\ &\approx -\frac{\partial p}{\partial x} dV \hat{e}_x. \end{aligned} \quad (6)$$

Compiling the x -, y - and z -components, and recalling the definition of the gradient, we get –

$$\text{Net pressure force on } V = -\frac{\partial p}{\partial x} dV \hat{e}_x - \frac{\partial p}{\partial y} dV \hat{e}_y - \frac{\partial p}{\partial z} dV \hat{e}_z = (-\nabla p) dV, \quad (7)$$

which tells us the net pressure force on the infinitesimal volume element. (Notational point: the text calls the volume δV .)

Besides the pressure force, we will also allow for gravitational forces on our volume element –

$$\text{Gravitational force on } V = m_V \mathbf{g} = \rho \mathbf{g} dV \quad (8)$$

where $m_V = \rho dV$ is the mass of fluid contained in V . Adding (8) to (7), we get

$$\text{Total force on } V = (-\nabla p + \rho \mathbf{g}) dV \quad (9)$$

This applied force has to be equal to the rate of change of the linear momentum of the fluid in V (the product of its mass and acceleration), that is,

$$\text{Mass} \times \text{acceleration of the fluid in } V = \rho dV \frac{D\mathbf{u}}{Dt} \quad (10)$$

where we have used the material derivative to express the acceleration of the fluid. We do this because the the forces described by (9) act on the fluid contained in V at a particular instant in time. Any changes in momentum effected by those forces are manifested in that parcel of fluid (what the book calls a ‘blob’) as it moves. The material derivative is thus relevant because it is the derivative that follows the fluid.

Equating (10) and (9), we get the remaining equations of motion for an ideal fluid,

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho} \nabla p + \mathbf{g}. \quad (11)$$

In general, (11) represents three equations, one for each dimension. They are nonlinear partial differential equations. Together, (3) and (11) are called *Euler’s equations* of motion for an ideal fluid.

(We didn’t use the conservation of energy in deriving Euler’s equations. Energy will become interesting when we talk about viscous fluids.)

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 4 - More about ideal fluids

Reading: Acheson, chapter 1.

In the last lecture we derived Euler's equations of motion for ideal fluids. (Recall that ideal fluids are incompressible, with constant density, and inviscid.) The Euler equations were –

$$\begin{aligned} \nabla \cdot \mathbf{u} &= 0 \\ \frac{D\mathbf{u}}{Dt} &= \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p + \mathbf{g}. \end{aligned} \quad (1)$$

We can rewrite the gravitational acceleration as $\mathbf{g} = -\nabla\chi$, where χ is a scalar potential, because the gravitational force is conservative. (In the usual coordinate arrangement where z is positive upward, $\chi = gz$.) Using this, the second equation in (1) becomes

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla \left(\frac{p}{\rho} + \chi \right) \quad (2)$$

where we have also used $\rho = \text{const.}$ Now, we apply the identity A.9 from the book to the velocity vector \mathbf{u} ,

$$(\mathbf{u} \cdot \nabla)\mathbf{u} = (\nabla \times \mathbf{u}) \times \mathbf{u} + \frac{1}{2}\nabla(\mathbf{u} \cdot \mathbf{u}).$$

(Notational point: for cross products, the text uses a caret ' \wedge ' rather than ' \times ', and what is written \mathbf{f}^2 in the text actually means $\mathbf{f} \cdot \mathbf{f} = |\mathbf{f}|^2$, the squared magnitude of the vector \mathbf{f} .) Inserting this into (2), we get

$$\frac{\partial \mathbf{u}}{\partial t} + (\nabla \times \mathbf{u}) \times \mathbf{u} = -\nabla \overbrace{\left(\frac{p}{\rho} + \frac{1}{2}\mathbf{u} \cdot \mathbf{u} + \chi \right)}^{\equiv H} = -\nabla H \quad (3)$$

where the quantity H starts to look familiar.

(Significantly, the two equations of (1) (actually four equations, since the second equation can be written as three equations, one for each velocity component) differ in that the first equation imposes a condition on the velocity field that is true for all times t , while the equations involving $\partial \mathbf{u} / \partial t$ describe the time *evolution* of the velocity field \mathbf{u} .)

1 Bernoulli's theorems

The Bernoulli streamline theorem. *In a steady flow of an ideal fluid, H is constant along streamlines.*

To show this, set $\partial \mathbf{u} / \partial t$ to zero in (3) since the flow is steady. We then get

$$(\nabla \times \mathbf{u}) \times \mathbf{u} = -\nabla H. \quad (4)$$

From the definition of the gradient, the rate of change of H in a given direction is equal to the dot product of ∇H with the unit vector in that direction. The unit vector in the direction of a streamline is $\hat{e}_{\mathbf{u}} = \mathbf{u} / |\mathbf{u}|$. Taking the dot product of $\hat{e}_{\mathbf{u}}$ with both sides of (4), we get

$$0 = \frac{\mathbf{u}}{|\mathbf{u}|} \cdot (\nabla H) = \hat{e}_{\mathbf{u}} \cdot \nabla H.$$

The left hand side is zero because $(\nabla \times \mathbf{u}) \times \mathbf{u}$ is perpendicular to \mathbf{u} (and $\nabla \times \mathbf{u}$) by the definition of the cross product, and the dot product of perpendicular vectors is zero. Thus, the rate of change of H in the \mathbf{u} direction is zero, i.e. H is constant along streamlines in a steady flow of an ideal fluid.

The Bernoulli theorem for irrotational flow. *In a steady, irrotational flow of an ideal fluid, H is constant everywhere.*

This theorem makes use of the definition of an irrotational flow, which is –

$$\text{In an irrotational flow, } \nabla \times \mathbf{u} = 0.$$

In steady flow the time derivative term in (3) goes to zero, and in an irrotational flow the term $(\nabla \times \mathbf{u}) \times \mathbf{u}$ goes to zero, leaving us with

$$0 = -\nabla H, \text{ i.e. } H \text{ is a constant everywhere.}$$

In your undergrad fluids class you probably used some form of these theorems until you practically couldn't see straight. Here we'll emphasize the assumptions that underlie these theorems. Both theorems require ideal fluids and steady flow, and the second theorem (which is probably the more familiar one) additionally requires that the flow be irrotational.

2 Vorticity

We have now encountered the *vorticity*, which is defined as –

$$\boldsymbol{\omega} = \nabla \times \mathbf{u}. \tag{5}$$

In the last section we defined an *irrotational* flow as one where the vorticity was zero, i.e. $\nabla \times \mathbf{u} = 0$. Logically enough, the vorticity is in some sense a measure of the local rotation in the flow. We emphasize 'local' because vorticity is distinct from fluid rotation in the global sense. The book illustrates the distinction very effectively (see particularly Figs. 1.6 and 1.8).

We derive the equation for the time evolution of the vorticity as follows. First, rewrite (3) as

$$\frac{\partial \mathbf{u}}{\partial t} + \boldsymbol{\omega} \times \mathbf{u} = -\nabla H.$$

Now take the curl of both sides. We can write

$$\nabla \times \frac{\partial \mathbf{u}}{\partial t} = \frac{\partial(\nabla \times \mathbf{u})}{\partial t} = \frac{\partial \boldsymbol{\omega}}{\partial t}$$

where we can switch the order of the time derivative and the curl, because the time derivative has nothing to do with the spatial variables and the curl has nothing to do with t . Also, the curl of a gradient is zero (identity A.2 in the text), so we get

$$\frac{\partial \boldsymbol{\omega}}{\partial t} + \nabla \times (\boldsymbol{\omega} \times \mathbf{u}) = 0. \tag{6}$$

Now we handle the $\nabla \times (\boldsymbol{\omega} \times \mathbf{u})$ term by using identity A.6 in the book, namely

$$\nabla \times (\boldsymbol{\omega} \times \mathbf{u}) = (\mathbf{u} \cdot \nabla)\boldsymbol{\omega} - (\boldsymbol{\omega} \cdot \nabla)\mathbf{u} + \boldsymbol{\omega}(\nabla \cdot \mathbf{u}) - \mathbf{u}(\nabla \cdot \boldsymbol{\omega}).$$

We can simplify this. The third term on the right hand side is zero because the fluid is incompressible. The fourth term is zero because the divergence of a curl is zero (identity A.3). Inserting into (6), we get the *vorticity equation* –

$$\frac{\partial \boldsymbol{\omega}}{\partial t} + (\mathbf{u} \cdot \nabla) \boldsymbol{\omega} = \frac{D\boldsymbol{\omega}}{Dt} = (\boldsymbol{\omega} \cdot \nabla) \mathbf{u}. \quad (7)$$

Interestingly, this equation for the time evolution of the vorticity doesn't include the pressure or gravity. (Gravity is an example of a body force, and disappears from the vorticity equation because it is conservative, and can thus be written as the gradient of a potential. Not all body forces are conservative - if nonconservative body forces were present, they couldn't be eliminated from the vorticity equation in this way.)

Interesting results can be derived from the vorticity equation if we consider a two-dimensional flow, where $\mathbf{u} = (u(x, y, t), v(x, y, t), 0)$. In such a case, the vorticity vector $\boldsymbol{\omega} = (0, 0, \omega_z)$, so that

$$(\boldsymbol{\omega} \cdot \nabla) \mathbf{u} = \omega_z \frac{\partial \mathbf{u}}{\partial z} = 0.$$

Thus, the vorticity equation for the two-dimensional flow of an ideal fluid is

$$\frac{D\omega_z}{Dt} = \frac{\partial \omega_z}{\partial t} + (\mathbf{u} \cdot \nabla) \omega_z = 0. \quad (8)$$

We can then make the following statement.

- *In two-dimensional flow of an ideal fluid, subject only to conservative body forces, the vorticity, ω_z , of each individual fluid element is conserved.*

Further, if the flow is steady, then (8) reduces to $(\mathbf{u} \cdot \nabla) \omega_z = 0$, i.e. ω_z doesn't change in the direction of \mathbf{u} . So we can also say:

- *In steady, two-dimensional flow of an ideal fluid subject to conservative body forces, the vorticity is constant along streamlines.*

3 Circulation

Two-dimensional flows are, strictly speaking, impermissible in nature, but certain systems can be pretty well approximated as two-dimensional, for example a thin cross-section from a long wing. The book includes a thorough discussion about the flow past a wing.

It will be useful to apply the idea of *circulation*, defined as –

$$\Gamma = \int_C \mathbf{u} \cdot d\mathbf{x}. \quad (9)$$

Stokes's theorem, applied to \mathbf{u} , is clearly relevant here –

$$\int_S (\nabla \times \mathbf{u}) \cdot \mathbf{n} dS = \int_C \mathbf{u} \cdot d\mathbf{x}.$$

In particular, from Stokes's theorem, the circulation, Γ , has to be zero for curves C that enclose surfaces S on which flows are irrotational. So, if we can approximate flow around a wing as two-dimensional, and if we assume that the surrounding fluid is ideal and irrotational, then the flow around a wing appears to have zero circulation.

But – we have to be careful in applying Stokes's theorem. If we draw a curve C around a wing, then the enclosed surface S includes the wing, and we obviously can't say that the flow is irrotational in the wing.

We'll pick up on the implications of this in the next lecture.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 5 - Limitations of ideal fluid theory / role of viscosity

Reading: Acheson, chapter 1, chapter 2 (through §2.3).

In the last lecture we introduced the concept of circulation, which we defined as –

$$\Gamma = \int_C \mathbf{u} \cdot d\mathbf{x}$$

where C is a closed curve. The circulation around an airfoil explains the lift generated. Consider the airfoil in Fig. 1, surrounded by a curve C , along which we will compute the circulation in the direction indicated. Assume that the mean airflow is horizontal, from left to right, in the reference

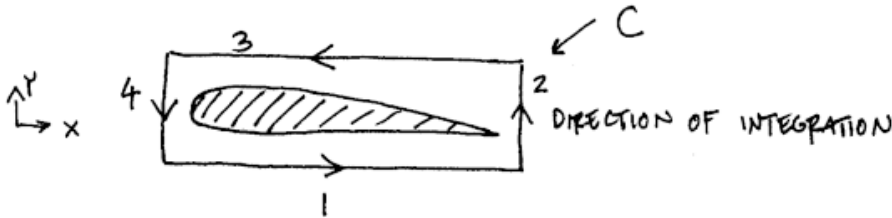


Figure 1: Computation of the circulation around a wing.

frame where the wing is stationary.

To compute the circulation, first assume that the sides labeled 2 and 4 are far enough away from the wing that the flow has no vertical component there, so $\mathbf{u} \cdot d\mathbf{x} = 0$ on those sides. Also, because C is a rectangle, sides 1 and 3 have the same length, but on side 1 we integrate in the positive x -direction and on side 3 we integrate in the negative x -direction. The circulation around C then becomes

$$\begin{aligned} \Gamma &= \overbrace{\int_{\text{side 1}} u \, dx} + \overbrace{\int_{\text{side 3}} -u \, dx} \\ &= \int (\overbrace{u}^{\text{side 1}} - \overbrace{u}^{\text{side 3}}) \, dx. \end{aligned} \quad (1)$$

What do we know about the values of u above and below the wing? We can assume that the flow is steady and irrotational, and that gravity is negligible. By the (second) Bernoulli theorem, $p + \frac{1}{2}\rho|\mathbf{u}|^2$ is constant everywhere. For the wing to have lift, the pressure on the lower surface has to be greater than that on the upper surface, which then means that the air speed below the wing (along side 3 of C) has to be less than the air speed above the wing (side 1 of C). From (1), this means that the circulation, Γ , is negative (for the direction of integration defined in Fig. 1).

Bernoulli's theorems are formulated for ideal fluids, and while real fluids aren't ideal, ideal fluid theory does a good job of predicting lift for airfoils at small angles of attack. (At high angles of attack, ideal fluid theory fails rapidly, as in Fig. 1.11 in the text.) We have shown that the lift of an airfoil depends on the circulation around it. Let's think a little bit more about the circulation being nonzero.

Specifically:

- We have assumed that the mean flow in which we've immersed our wing is irrotational (has zero vorticity), that the flow around the wing can be treated as two-dimensional. These are perfectly valid assumptions.

- By Stokes's theorem, we showed that the circulation around a curve C is zero if the vorticity is zero over the entire surface S bounded by C .
- By considering the vorticity equation in the last lecture, we showed that for two-dimensional flows of ideal fluid subject to conservative body forces, the vorticity of each individual fluid element is conserved.

Given all this, we might reasonably think:

- It appears that vorticity correlates with circulation, and the vorticity in the flow surrounding the wing is zero.
- So how does the circulation around the wing arise?

1 Viscosity

To explain this properly, we have to consider what happens when a wing, initially at rest, is set into motion suddenly. (Described in §1.1 in the text.) It turns out that as the wing is set into motion, a vortex (with, of course, nonzero vorticity) forms at the trailing edge of the wing. As we will (hopefully) cover later, the circulation contained in the vortex is equal and opposite to the circulation around the wing. The vortex trails further and further behind the wing as time evolves. When we look at a wing in steady flow, we can assume that the vortex is, effectively, infinitely far behind the wing, so the wing can be considered to be surrounded by irrotational flow. However, the circulation around the wing, which balances the circulation in the vortex, remains.

Here is a subtle point: while the *existence* of circulation around an airfoil is permissible by ideal fluid theory, we can't explain the *origin* of the circulation unless the *viscosity* of the fluid is considered.

Viscosity can be a tricky concept because there are major qualitative differences in fluid flows with very small viscosity and those with no viscosity. That is, the behavior of a fluid with very small viscosity can be vastly different from the behavior of a fluid with zero viscosity.

1.1 Viscous forces

One of the properties that define an ideal fluid is that the only force acting across any surface element in the fluid is the pressure, which acts in a direction normal to the plane. The simplest explanation of viscous fluids is that they have an additional force that acts tangential to a given surface element. This force depends on the velocity gradient across the surface element. A fast layer of fluid wants to drag a slow layer along with it, and the slow layer wants to slow down the fast layer. (Fig. 2.) As shown in the figure, while pressure is a normal force (meaning its direction),

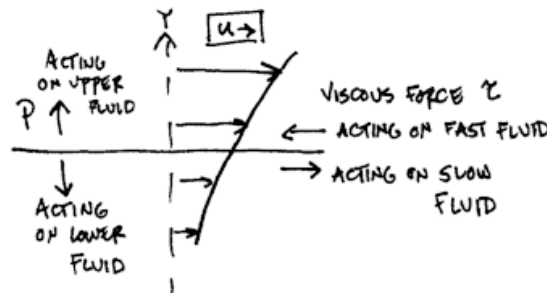


Figure 2: Pressure forces and viscous shear forces across a surface in a fluid.

viscosity imposes shear forces¹. Specifically, *viscous fluids resist shear*, i.e. they oppose velocity gradients.

Many of the qualitative differences between viscous and inviscid fluids arise because viscous fluid flows have *boundary layers* near solid surfaces. Boundary layers arise because of the *no-slip* condition for viscous fluids, which holds that a fluid element in contact with a solid boundary must have the same velocity as the boundary. (Essentially, this is because momentum is transferred across a fluid-solid interface similarly to the way momentum is transferred across a fluid-fluid interface.) The result of the no-slip condition is that while a flow can appear to be happily streaming past a body, say a wing or a car or something, at the actual solid surface the fluid and the body have the same velocity. One clear manifestation of this is the way you can't clean off a dusty car just by driving fast.

In this class we will only be concerned with *Newtonian* fluids, where the viscous shear stress, τ , is proportional to the velocity gradient. In Fig. 1, this means that –

$$\tau = \mu \frac{du}{dy}$$

where μ is called the coefficient of dynamic viscosity.

¹There are also normal viscous forces, which we'll describe later, but the shear force is the one that's more intuitive.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 6 - Basic viscous flow ideas

Reading: Acheson, chapter 2 (through §2.3).

In the last lecture we introduced the concept of *viscosity*. The most intuitively understandable property of viscous fluids is that they resist shear. For example, given a parallel flow of viscous fluid with a velocity gradient perpendicular to the flow direction, viscous shear forces will be present that act to make the flow speed uniform, by simultaneously accelerating slower fluid and decelerating fast fluid (Fig. 1).

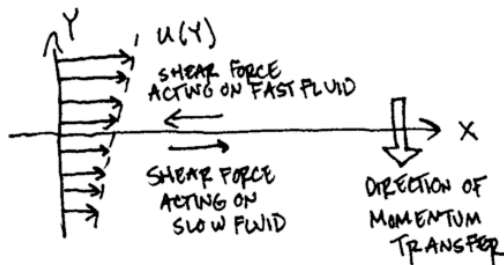


Figure 1: Viscous shear forces in a parallel fluid flow.

Viscosity is a diffusive property. In particular, owing to viscosity, momentum in a fluid diffuses from regions of high momentum to regions of low momentum. (The rate of momentum loss is equal to a negative force, and the rate of momentum increase is equal to a positive force.) As such, viscous diffusion of momentum is analogous to the diffusion of heat or species concentration. We will also see that viscosity is responsible for the diffusion of vorticity.

In the last lecture, viscosity arose in our discussion through the notion of the *boundary layer*. Specifically, it was pointed out that while the flow around an airfoil could be treated for the most part using ideal fluid theory, the generation of circulation around an airfoil – by which the wing generates lift – can only be explained by considering the viscosity of the fluid. In the case of the wing, as with many flow problems, the influence of viscosity is confined to a thin boundary layer near the solid surface of the wing. The boundary layer arises because of the *no-slip* condition for viscous flows, which states that at a solid surface, the fluid must have the same velocity as the surface. The result is a boundary layer region in which the flow makes the adjustment from the essentially inviscid main flow to the velocity values at the solid body. If the viscosity is small, this boundary layer can be very thin.

Figure 1.11 in the book shows that ideal fluid theory is quite valid for a wing until its angle of attack exceeds a critical value. The failure of the ideal fluid theory arises when the boundary layer ceases to be thin and instead affects a substantial portion of the flow. This phenomenon is known as *boundary layer separation*, and occurs when the boundary layer experiences a strong increase in pressure along the solid boundary. Boundary layer separation is responsible for significant qualitative differences in flow properties between ideal fluids and those with even very small viscosity. The classic example of drag on a sphere (e.g. why a dimpled golf ball has less drag than a smooth ball) is fundamentally tied up with boundary layer separation. (More on this later.)

1 Equations of motion for viscous fluids

We'll have a look at the equations of motion for viscous fluids here, go over some sample problems, and defer the detailed derivation of the equations for later. We will restrict our attention to

Newtonian fluids, where the viscous shear stress is directly proportional to the velocity gradient, that is (in the coordinate system of Fig. 1)

$$\tau = \mu \frac{du}{dy}, \text{ where } \mu \text{ is called the } \textit{coefficient of dynamic viscosity}.$$

This shear stress τ has the units of pressure (force/area), so the viscous shear force, F_μ , acting across a surface element in a flow with area A is

$$F_\mu = \tau \cdot A = \mu \frac{du}{dy} A.$$

The equations of motion for a Newtonian fluid with constant density, ρ , and constant viscosity, μ , are

$$\begin{aligned} \nabla \cdot \mathbf{u} &= 0 \\ \frac{D\mathbf{u}}{Dt} &= \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\frac{1}{\rho} \nabla p + \frac{\mu}{\rho} \nabla^2 \mathbf{u} + \mathbf{g}. \end{aligned} \quad (1)$$

These are known as the *Navier-Stokes equations*, and are the same as the Euler equations we've been discussing except for the viscous term, $(\mu/\rho)\nabla^2 \mathbf{u}$.

We showed earlier that the Laplacian operator, ∇^2 , could be interpreted in light of our understanding of the divergence, because the Laplacian is just the divergence of a gradient. That is, given a scalar quantity, ϕ ,

$$\nabla^2 \phi = \nabla \cdot (\nabla \phi).$$

To interpret the viscous term in the Navier-Stokes equations, first observe that the second equation in (1) is really three equations, one for each of the velocity components. Let's consider Cartesian coordinates, so $\mathbf{u} = (u, v, w)$, and write the equation for the u -component:

$$\frac{\partial u}{\partial t} + (\mathbf{u} \cdot \nabla)u = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \frac{\mu}{\rho} \nabla^2 u, \quad (2)$$

where particular care has to be taken with the second term on the left hand side. The gravitational term has been assumed to lie in the z -direction, as is conventional in Cartesian coordinates.

Let's now take a (somewhat crude) look at the Laplacian in (2), $\nabla^2 u = \partial^2 u / \partial x^2 + \partial^2 u / \partial y^2 + \partial^2 u / \partial z^2$. As with our discussions of the divergence, start with the familiar infinitesimal cubic volume element V (Fig. 2). We want to calculate the viscous forces on each face. Consider first the

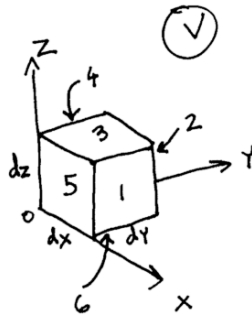


Figure 2: Our usual cubic volume element.

shear forces. To evaluate these, we need the velocity derivatives $\partial u / \partial y$ and $\partial u / \partial z$, since the fluid

is Newtonian. Specifically, on faces 2 and 5, where the normal vector is parallel to the y -axis, we need $\partial u/\partial y$, and on faces 3 and 6, where the normal is parallel to the z -axis, we need $\partial u/\partial z$. As usual, on each of these faces the derivative components of interest will be assumed to be constant.

Figure 3 shows us how to interpret the shear terms in the Laplacian, $\partial^2 u/\partial y^2$ and $\partial^2 u/\partial z^2$. Fig. 3a illustrates the variation of the u -component of the velocity in the y -direction, which is

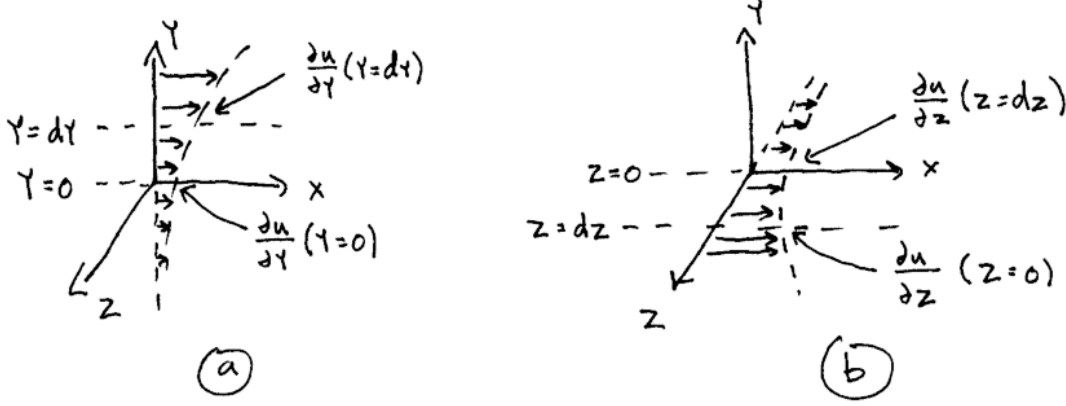


Figure 3: Interpretation of the viscous shear terms.

relevant to faces 2 and 5. Consider face 2, i.e. the $y = dy$ surface of V . The faster fluid at $y > dy$ imposes a force in the positive x -direction on the fluid inside V , represented by

$$\text{Viscous shear force on face 2} = \mu \left[\frac{\partial u}{\partial y}(y = dy) \right] dx dz.$$

Meanwhile, on face 5 ($y = 0$), the slower fluid at $y < 0$ imposes a force in the negative x direction on the fluid inside V :

$$\text{Viscous shear force on face 5} = -\mu \left[\frac{\partial u}{\partial y}(y = 0) \right] dx dz.$$

The net viscous shear force on faces 2 and 5 is thus, with some manipulation,

$$\begin{aligned} \text{Net viscous shear force on faces 2 and 5} &= \mu \left[\frac{\partial u}{\partial y}(y = dy) - \frac{\partial u}{\partial y}(y = 0) \right] dx dz \\ &= \mu \frac{\frac{\partial u}{\partial y}(y = dy) - \frac{\partial u}{\partial y}(y = 0)}{dy} dx dy dz \\ &= \mu \frac{\partial^2 u}{\partial y^2} dV \end{aligned} \quad (3)$$

Since (2) describes the rate of change of the velocity component u , we need to divide the mass out of (3). The mass contained in V is just $m_V = \rho dV$, so, dividing this out of (3), we get the term

$$\frac{\mu}{\rho} \frac{\partial^2 u}{\partial y^2}$$

consistent with (2). The variation of the u -component of the velocity in the z -direction (Fig. 3b) can be described similarly, and gives rise to the term $\frac{\mu}{\rho} \frac{\partial^2 u}{\partial z^2}$.

The variation of the u -component in the x -direction leads to a similar term, $\mu \frac{\partial^2 u}{\partial x^2}$, but in this case the viscous effect is not a *shear* effect, but instead a *normal* effect. This arises because viscous

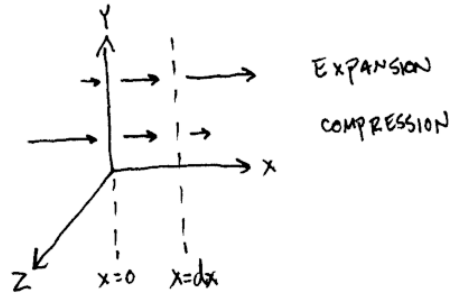


Figure 4: Interpretation of the normal shear term.

fluids resist not only resist shearing, but also resist expansion and compression (think of pulling apart or squeezing, say, a blob of caramel). An expansion is represented by $(\partial u / \partial x) > 0$, while a compression would have $\partial u / \partial x < 0$ (Fig. 4). It is somewhat unintuitive that compressions and expansions should be described mathematically in a similar fashion to shearing, but this turns out to be the case. That is, $\mu \frac{\partial u}{\partial x}$ describes a normal viscous force on a face, just as $\mu \frac{\partial u}{\partial y}$ and $\mu \frac{\partial u}{\partial z}$ described shear viscous forces.

Consider face 1 of our volume V , corresponding to the $x = dx$ surface in Fig. 4. In the expansion case, $\partial u / \partial x > 0$ across $x = dx$, and the faster fluid at $x > dx$ will act to pull on, or accelerate, the slower fluid in V . On face 4, in the expansion case, the slow fluid at $x < 0$ imposes a drag on the fluid in V . The net result is (skipping some steps that are exactly analogous to the development of (3)):

$$\text{Net normal force on faces 1 and 4} = \mu \frac{\partial^2 u}{\partial x^2} dV.$$

So, the viscous term in (2) breaks down as

$$\frac{\mu}{\rho} \nabla^2 u = \frac{\mu}{\rho} \left[\overbrace{\frac{\partial^2 u}{\partial x^2}}^{\text{Normal force on faces 1,4}} + \overbrace{\frac{\partial^2 u}{\partial y^2}}^{\text{Shear force on faces 2,5}} + \overbrace{\frac{\partial^2 u}{\partial z^2}}^{\text{Shear force on faces 3,6}} \right]. \quad (4)$$

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 7 - The Reynolds number/some viscous flow examples

Reading: Acheson, §2.2–2.4.

In the last lecture we wrote down the Navier-Stokes equations of motion for incompressible, viscous fluids:

$$\begin{aligned} \nabla \cdot \mathbf{u} &= 0 \\ \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} &= -\frac{\nabla p}{\rho} + \frac{\mu}{\rho} \nabla^2 \mathbf{u} + \mathbf{g}. \end{aligned} \quad (1)$$

The first of these equations describes the incompressibility; the remaining equations (one for each spatial dimension) describe the evolution of the velocity field (and are sometimes referred to as the momentum equations). These equations differ from the the Euler equations discussed earlier by the presence of the viscous term $\frac{\mu}{\rho} \nabla^2 \mathbf{u}$. As we discussed in the last lecture, this term covers both viscous shear forces and viscous normal forces.

1 The Reynolds number and kinematic viscosity

The *Reynolds number* quantifies the importance of viscosity in a given fluid flow. It is defined as

$$Re \equiv \frac{\rho U L}{\mu}, \quad (2)$$

where U is a characteristic velocity in the flow, L is a characteristic length scale, and ρ and μ are the density and dynamic viscosity of the fluid, respectively.

To see how to interpret the Reynolds number, let's look at where it comes from. We will want to rewrite the momentum equation in (1) in non-dimensional form. Define

$$x_i^* = \frac{x_i}{L}, \quad u_i^* = \frac{u_i}{U}, \quad t^* = \frac{t}{L/U}, \quad (3)$$

which are the non-dimensional forms of these variables. It is important to use the non-dimensional form of the *operators* as well, i.e.

$$\nabla^* = L \nabla, \quad \frac{\partial}{\partial t^*} = \frac{L}{U} \frac{\partial}{\partial t}. \quad (4)$$

Now use (3) and (4) in (1), and we get

$$\frac{U^2}{L} \frac{\partial \mathbf{u}^*}{\partial t^*} + \frac{U^2}{L} (\mathbf{u}^* \cdot \nabla^*) \mathbf{u}^* = -\frac{1}{L} \frac{\nabla^* p}{\rho} + \frac{U}{L^2} \frac{\mu}{\rho} \nabla^{*2} \mathbf{u}^* + \mathbf{g}.$$

Now multiply (L/U^2) through on both sides:

$$\begin{aligned} \frac{\partial \mathbf{u}^*}{\partial t^*} + \overbrace{(\mathbf{u}^* \cdot \nabla^*) \mathbf{u}^*}^{\text{'inertial term'}} &= -\nabla^* p^* + \frac{\mu}{\rho U L} \nabla^{*2} \mathbf{u}^* + \frac{L}{U^2} \mathbf{g} \\ &= -\nabla^* p^* + \underbrace{\frac{1}{Re} \nabla^{*2} \mathbf{u}^*}_{\text{'viscous term'}} + \frac{L}{U^2} \mathbf{g}, \end{aligned} \quad (5)$$

where we have written $p^* = p/(\rho U^2)$, which is the common non-dimensionalization for the pressure.

If we have chosen the parameters U and L correctly for the problem, then the various non-dimensional terms involving u^* and its derivatives in (5) should be of order 1. In that case, the value of the Reynolds number tells us the relative importance of the inertial and viscous terms –

$$O(Re) = \frac{|\text{inertial term}|}{|\text{viscous term}|}.$$

Thus, a high Reynolds number means viscous effects are less important, and a low Reynolds number means viscous effects are very important. For ideal fluids, then, it is required that the Reynolds number be very large. However, just because the Reynolds number in a flow is large doesn't mean that we can automatically assume that ideal fluid theory applies. This is because *turbulence* develops at high Re , and turbulence complicates matters immensely.

You will have noticed that in the Navier-Stokes equations, viscosity appears divided by the density, so we always see μ/ρ . This is called the *kinematic viscosity*, $\nu \equiv \mu/\rho$, and is the relevant viscosity in most fluid problems. The kinematic viscosity is anti-intuitive in some ways. For example, the dynamic viscosity, μ , of water is about 55 times that of air, which is probably consistent with your intuition. However, the kinematic viscosity of air is about 15 times that of water, so in that sense air is *more* viscous than water.

2 Plane Poiseuille flow

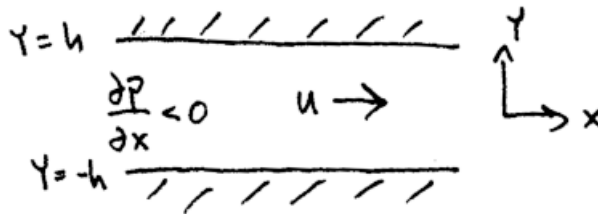


Figure 1: Plane Poiseuille flow.

Consider the system of Fig. 1, consisting of the flow of a viscous, incompressible fluid between two fixed, parallel plates, where the flow is driven by a constant pressure gradient $\partial P/\partial x$. (This is known as plane Poiseuille flow.) We want to know the velocity field, which in general is

$$\mathbf{u} = (u(x, y, z, t), v(x, y, z, t), w(x, y, z, t)).$$

The first thing we assume is that the flow is steady, because the pressure gradient driving the flow is constant. We will also assume that the velocity doesn't vary in the x - or z -directions. To justify this, consider that to an observer who's free to move around in the $x-z$ plane (for fixed values of y), all points in the plane look the same – the pressure gradient is constant, and the boundaries don't vary with x or z . We can also assume that the w -component of the velocity is zero, because there is no pressure gradient or gravitational force that could potentially drive the flow in the z -direction. This leads to

$$\mathbf{u} = (u(y), v(y), 0)$$

and we haven't needed the Navier-Stokes equations yet.

Now we'll apply the Navier-Stokes equations. Because the flow is incompressible,

$$\nabla \cdot \mathbf{u} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0.$$

The first term $\partial u/\partial x$ is zero because u is not a function of x , and the third term $\partial w/\partial z$ is zero because $w = 0$, so this means that $\partial v/\partial y$ is also zero. We already have $v = v(y)$, so $\partial v/\partial y = 0$ means that v is constant everywhere. But $v = 0$ at $y = \pm h$ because the fixed walls are there, so $v = 0$ everywhere. (Observe that this also means we are ignoring gravity.) Thus,

$$\mathbf{u} = (u(y), 0, 0)$$

We are left with the following terms in the Navier-Stokes equation for the u -component of velocity:

$$0 = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \frac{\partial^2 u}{\partial x^2} = -C + \nu \frac{\partial^2 u}{\partial x^2}$$

where C is a constant (because the pressure gradient and density are constant). Integrating once,

$$Cy + C_1 = \nu \frac{\partial u}{\partial y}.$$

To find C_1 , note that the u profile needs to be symmetric around $y = 0$ (why?), so $\partial u/\partial y = 0$ at $y = 0$, and $C_1 = 0$. Integrating again, we get

$$\frac{1}{2}Cy^2 + C_2 = \nu u.$$

The boundary condition on u is that $u = 0$ at $y \pm h$. This allows us to evaluate C_2 . The final result is

$$u = \frac{C}{2\nu}(y^2 - h^2).$$

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 8 - More viscous flow examples

Reading: Acheson, §2.3–2.5.

In the last class we illustrated some basic viscous flow ideas by finding the velocity field for the steady flow of viscous, incompressible fluid driven by a constant pressure gradient between two parallel plates (plane Poiseuille flow). In this lecture we will look at a more sophisticated example. The added sophistication doesn't arise particularly in the physics of the problem, but rather in the mathematical tools necessary for solving the differential equations.

1 Flow due to an impulsively moved plane boundary

(This example is from the book. We'll work it out in full detail here.)

Consider the system in Fig. 1. A viscous, incompressible fluid occupies the region $y > 0$, which is bounded by a solid wall at $y = 0$. The wall is initially at rest, but at $t > 0$ the wall has velocity U in the x -direction. The flow is not pressure-driven. We want to compute the velocity field $\mathbf{u}(\mathbf{x}, \mathbf{t}) = (u, v, w)$ for $t > 0$.

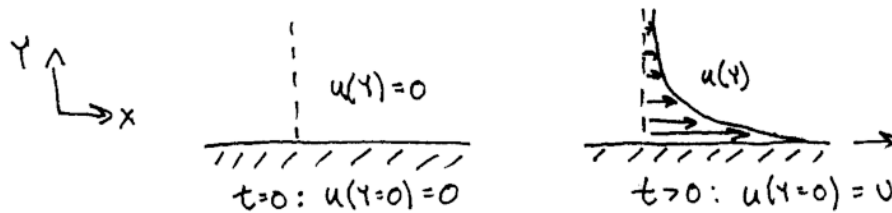


Figure 1: An impulsively moved plane boundary.

As usual we want to simplify the problem as much as possible before looking at equations. First of all, the geometry of the problem doesn't vary in the x - and z -directions, so we assume that the velocity components are only functions of y . Also, there is nothing driving the flow in the z -direction, so we assume that $w = 0$. This gives us $\mathbf{u} = (u(y, t), v(y, t), 0)$.

Because the flow is incompressible, $\nabla \cdot \mathbf{u} = \partial u / \partial x + \partial v / \partial y = 0$. We have $\partial u / \partial x = 0$ since $u = u(y, t)$, so $\partial v / \partial y = 0$. We know $v = v(y, t)$, so $\partial v / \partial y = 0$ means v is constant everywhere – but we have as a boundary condition that $v = 0$ at $y = 0$, so finally, we get $v = 0$ everywhere. Thus

$$\mathbf{u}(\mathbf{x}, \mathbf{t}) = (u(y, t), 0, 0). \quad (1)$$

(This is a *plane parallel shear flow*.)

The Navier-Stokes momentum equation is:

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{\nabla p}{\rho} + \nu \nabla^2 \mathbf{u} + \mathbf{g}.$$

We are only interested in the u -component of this equation, which, for the velocity field of (1), works out as

$$\frac{\partial u}{\partial t} = \nu \frac{\partial^2 u}{\partial y^2}. \quad (2)$$

We have eliminated the pressure gradient term because the flow is not pressure-driven. This equation is called the one-dimensional diffusion equation, and arises with diffusive properties such as heat. (The thermodynamic analogue of this problem would be if our fluid had initially constant temperature, and suddenly the temperature of the boundary was changed.)

The conditions on this problem are:

$$\begin{aligned} \text{Initial condition: } & u(y, 0) = 0 \text{ for } y > 0. \\ \text{Boundary conditions: } & u(0, t) = U \text{ and } u(\infty, t) = 0 \text{ for } t > 0. \end{aligned} \tag{3}$$

We are going to look for something called a *similarity solution*. There is a lot of lore about similarity solutions in the mathematical literature, but unfortunately there seem to be no general rules about when you can and can't look for similarity solutions and even if there were, the math would be way too complicated for us. Instead, we'll rely on reasoning.

Let's consider what's happening in this system. Initially the fluid and the wall are at rest (Fig. 1). Suddenly, the wall is set in motion at velocity $u = U$. Because the fluid is viscous, the fluid in contact with the wall also moves at speed U by the no-slip condition. This establishes a velocity difference, or gradient, with the layer of slower fluid immediately above. The resulting viscous shear stress acts to accelerate that layer of slower fluid. As its velocity increases, this layer, in turn, acts to accelerate the slower fluid above it, and on and on. At any time in the process, we get a velocity distribution that looks something like that on the right in Fig. 1. Basically, the moving wall introduces momentum to the system, which gets diffused upward by the fluid viscosity as time proceeds.

What can we say about the shape of the velocity profile as time proceeds? One thing we can assume is that U is a scale factor for the velocity field, so a change in U would have a direct, linear effect on the velocity values $u(y)$. For example, if we double U , then we would just double all of the velocity values in the fluid for a given t .

Another thing we notice is that the process by which the different 'layers' of fluid continually accelerate slower fluid above them has no stopping point, because the fluid domain is infinite in the positive y -direction. So there is no fixed length scale set by the boundary conditions in the y -direction. This suggests that if we look at the velocity profile at successively later times, the shape will be the same but the profile will just be continually stretched in the y -direction. What is the stretching factor for the profile height?

The text describes one way to find this 'stretching' (or normalizing) factor, δ . Another way is to use dimensional analysis. The height of the profile can only depend on the time, t , and the viscosity, ν . The only other parameter in the problem is U , and we've already reasoned that it just scales the velocity values. (Looking at the u profile in Fig. 1, this means that U scales the profile horizontally, and doesn't affect the height in y .) The viscosity, ν , has units of $(\text{length})^2/(\text{time})$, so the only way to form a variable, δ , with units of length from ν and t is to set $\delta = \sqrt{\nu t}$.

So we are looking for a velocity field of the form

$$u = U f(\eta), \text{ where } \eta = \frac{y}{\delta} = \frac{y}{\sqrt{\nu t}}.$$

By the chain rule, we have (using the notation $df/d\eta = f'(\eta)$)

$$\begin{aligned} \frac{\partial u}{\partial t} &= f'(\eta) \frac{\partial \eta}{\partial t} = -f'(\eta) \frac{y}{2\nu^{1/2}t^{3/2}}, \\ \frac{\partial u}{\partial y} &= f'(\eta) \frac{\partial \eta}{\partial y} = f'(\eta) \frac{1}{\nu^{1/2}t^{1/2}}, \text{ and} \\ \frac{\partial^2 u}{\partial y^2} &= f''(\eta) \frac{1}{\nu t}. \end{aligned}$$

Inserting into (2), we get

$$\begin{aligned} -f'(\eta) \frac{y}{2\nu^{1/2}t^{3/2}} &= f''(\eta) \frac{1}{t}, \text{ or} \\ -\frac{1}{2}\eta f' &= f''. \end{aligned} \quad (4)$$

Observe that this is a first-order differential equation, because one term involves the first derivative and the other term involves the second derivative. So we can define $g(\eta) = f'(\eta)$, and inserting into (4), we get

$$-\frac{1}{2}\eta g = \frac{dg}{d\eta}.$$

We can use the separation of variables to write this as

$$-\frac{1}{2}\eta d\eta = \frac{dg}{g}.$$

Integrating both sides, we get

$$-\frac{1}{4}\eta^2 + C = \ln(g),$$

where C is an arbitrary constant. Taking the exponential of both sides gives us

$$g = e^C e^{-\eta^2/4} = B e^{-\eta^2/4} = \frac{df}{d\eta}, \quad (5)$$

where $B = e^C$ and we have re-substituted $g = df/d\eta$.

Integrating (5) once, we get

$$f(\eta) = A + B \int_0^\eta e^{-s^2/4} ds. \quad (6)$$

We use the dummy variable of integration s , because we want to evaluate the integral at specific limits so we can impose boundary conditions, and there is no analytical expression for the integral of $e^{-s^2/4}$. Our boundary conditions were given in (3). Since the similarity function f was defined by $u = Uf(\eta)$, our conditions on f are $f(0) = 1$ and $f(\infty) = 0$. The condition $f(0) = 1$ is easy, because the integral in (6) goes to zero, so $A = 1$. To evaluate f at $\eta = \infty$, we exploit the fact that the definite integral

$$\int_0^\infty e^{-s^2/4} ds = \sqrt{\pi}$$

is found in integral tables. So for f to satisfy the boundary condition at $\eta = \infty$ requires that $B = -\frac{1}{\sqrt{\pi}}$. So finally, we get as our solution for u -

$$u(y, t) = U \left(1 - \frac{1}{\sqrt{\pi}} \int_0^\eta e^{-s^2/4} ds \right), \text{ where } \eta = \frac{y}{\sqrt{\nu t}}. \quad (7)$$

The velocity field defined by (7) basically looks like the one depicted in Fig. 1.

2 Diffusion of vorticity/flow with circular streamlines

We said before that viscosity was responsible for the diffusion of both momentum and vorticity. This can be seen explicitly by looking at the equation for the evolution of the vorticity field, $\omega(\mathbf{x}, t)$:

$$\frac{\partial \omega}{\partial t} + (\mathbf{u} \cdot \nabla) \omega = (\omega \cdot \nabla) \mathbf{u} + \nu \nabla^2 \omega, \quad (8)$$

in which the term $\nu \nabla^2 \omega$ describes the viscous diffusion of vorticity, analogously to the viscous term in the Navier-Stokes momentum equation.

To illustrate the viscous diffusion of vorticity, it is convenient to consider flows with circular streamlines. These are most naturally described in cylindrical coordinates. In particular, we are interested in velocity fields of the form

$$\mathbf{u} = u_\theta(r, t) \hat{e}_\theta. \quad (9)$$

The book gives the Navier-Stokes equations in cylindrical coordinates in §2.4. The one we're interested in is the equation for the time evolution of the u_θ velocity component, which is

$$\begin{aligned} \frac{\partial u_\theta}{\partial t} + (\mathbf{u} \cdot \nabla) u_\theta + \frac{u_r u_\theta}{r} &= -\frac{1}{\rho r} \frac{\partial p}{\partial \theta} + \nu \left(\nabla^2 u_\theta + \frac{2}{r^2} \frac{\partial u_r}{\partial \theta} - \frac{u_\theta}{r^2} \right), \\ \text{where } \mathbf{u} \cdot \nabla &= u_r \frac{\partial}{\partial r} + \frac{u_\theta}{r} \frac{\partial}{\partial \theta} + u_z \frac{\partial}{\partial z}, \\ \text{and } \nabla^2 &= \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2}{\partial \theta^2} + \frac{\partial^2}{\partial z^2}. \end{aligned}$$

For the velocity field of (9), we then have $(\mathbf{u} \cdot \nabla) u_\theta = 0$ and $\nabla^2 u_\theta = \frac{\partial^2 u_\theta}{\partial r^2} + \frac{1}{r} \frac{\partial u_\theta}{\partial r}$, which gives us

$$\frac{\partial u_\theta}{\partial t} = -\frac{1}{\rho r} \frac{\partial p}{\partial \theta} + \nu \left(\frac{\partial^2 u_\theta}{\partial r^2} + \frac{1}{r} \frac{\partial u_\theta}{\partial r} - \frac{u_\theta}{r^2} \right). \quad (10)$$

Consider the pressure term. Every other term in (10) is a function of r and t only, so we can write

$$\frac{\partial p}{\partial \theta} = P(r, t).$$

Integrating, we get

$$p = P(r, t) \theta + C(r, t)$$

where $C(r, t)$ is a constant of integration with respect to the variable θ , which means that it can vary with r and t . We can say that $P(r, t)$ has to be zero, because otherwise $p(\theta) \neq p(\theta + 2\pi)$, i.e. p would have multiple values at a given position. Thus p is not a function of θ , so (10) becomes

$$\frac{\partial u_\theta}{\partial t} = \nu \left(\frac{\partial^2 u_\theta}{\partial r^2} + \frac{1}{r} \frac{\partial u_\theta}{\partial r} - \frac{u_\theta}{r^2} \right). \quad (11)$$

We will use this to explore particular flows with circular streamlines in the next lecture.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 9 - Even more viscous flow examples

Reading: Acheson, §2.4–2.5.

In the last lecture, we began to consider flows with circular streamlines, for which the velocity fields are given by

$$\mathbf{u} = u_\theta(r, t) \hat{e}_\theta.$$

For such a flow, the Navier-Stokes equation describing incompressibility, $\nabla \cdot \mathbf{u} = 0$, is obviously satisfied. The Navier-Stokes equation for the time evolution of u_θ is

$$\frac{\partial u_\theta}{\partial t} = \nu \left(\frac{\partial^2 u_\theta}{\partial r^2} + \frac{1}{r} \frac{\partial u_\theta}{\partial r} - \frac{u_\theta}{r^2} \right). \quad (1)$$

Considering flows with circular streamlines is a convenient way to illustrate the viscous diffusion of vorticity. We'll work through one of the book examples, that of the viscous decay of a line vortex.

1 Viscous decay of a line vortex

Define a line vortex as

$$\mathbf{u} = \frac{\Gamma_0}{2\pi r} \hat{e}_\theta, \quad (2)$$

with Γ_0 constant. For a flow with circular streamlines, the vorticity is computed as

$$\omega = \nabla \times \mathbf{u} = \frac{1}{r} \frac{\partial(r u_\theta)}{\partial r}.$$

For the line vortex flow of (2), then, the vorticity works out as

$$\omega = \frac{1}{r} \frac{\partial(\Gamma_0/2\pi)}{\partial r} = \begin{cases} \text{undefined} & \text{if } r = 0 \\ 0 & \text{if } r > 0. \end{cases}$$

That is, there is a vorticity singularity at $r = 0$. If we set this line vortex as the initial condition, we want to know what happens to the flow as time proceeds.

Instead of proceeding by looking at the velocity, u_θ , the book chooses to proceed using the circulation:

$$\Gamma(r, t) = 2\pi r u_\theta(r, t), \quad (3)$$

where, obviously,

$$u_\theta(r, t) = \frac{\Gamma(r, t)}{2\pi r}.$$

Expressed as an evolution equation for the circulation, the Navier-Stokes equation (1) is thus

$$\frac{\partial \Gamma}{\partial t} = \nu \left(\frac{\partial^2 \Gamma}{\partial r^2} - \frac{1}{r} \frac{\partial \Gamma}{\partial r} \right), \quad (4)$$

using

$$\begin{aligned}\frac{\partial u_\theta}{\partial t} &= \frac{1}{2\pi r} \frac{\partial \Gamma}{\partial t}, \\ \frac{\partial u_\theta}{\partial r} &= \frac{1}{2\pi r} \frac{\partial \Gamma}{\partial r} - \frac{\Gamma}{2\pi r^2}, \text{ and} \\ \frac{\partial^2 u_\theta}{\partial r^2} &= \frac{1}{2\pi r} \frac{\partial^2 \Gamma}{\partial r^2} - \frac{1}{\pi r^2} \frac{\partial \Gamma}{\partial r} + \frac{\Gamma}{\pi r^3}.\end{aligned}$$

The initial condition is clear from the definition of the line vortex (2) and the expression for the circulation (3) –

$$\Gamma(r, t = 0) = \Gamma_0. \quad (5)$$

We also have the boundary condition that u_θ be finite at $r = 0$ and $t > 0$, namely

$$\Gamma(r = 0, t) = 0 \text{ for } t > 0. \quad (6)$$

This problem is very similar in form to the one about the impulsively moved plane boundary we talked about in the last lecture. As we did there, we will look for a similarity solution, of the form

$$\Gamma = \Gamma_0 f(\eta), \text{ where } \eta = r/\sqrt{\nu t}. \quad (7)$$

Plugging (7) into (4), we get

$$f' \frac{\partial \eta}{\partial t} = \nu \left[f'' \left(\frac{\partial \eta}{\partial r} \right)^2 + f' \frac{\partial^2 \eta}{\partial r^2} - \frac{1}{r} f' \frac{\partial \eta}{\partial r} \right], \quad (8)$$

using the notation $df/d\eta = f'$. For our definition of η , we have

$$\begin{aligned}\frac{\partial \eta}{\partial t} &= -\frac{r}{2\nu^{1/2}t^{3/2}}, \\ \frac{\partial \eta}{\partial r} &= \frac{1}{\sqrt{\nu t}}, \text{ and} \\ \frac{\partial^2 \eta}{\partial r^2} &= 0.\end{aligned}$$

Inserted in (8):

$$-f' \frac{r}{2\nu^{1/2}t^{3/2}} = \nu \left[f'' \frac{1}{\nu t} - \frac{1}{r} f' \frac{1}{\sqrt{\nu t}} \right].$$

Multiplying through by $\frac{r}{\nu} \sqrt{\nu t}$ on both sides:

$$\begin{aligned}-f' \frac{r^2}{2\nu t} &= f'' \frac{r}{\sqrt{\nu t}} - f', \text{ giving} \\ -f' \frac{\eta^2}{2} &= f'' \eta - f', \text{ or} \\ -f' \left(\frac{\eta}{2} - \frac{1}{\eta} \right) &= f''.\end{aligned} \quad (9)$$

As in the last lecture, we approach (9) first as a first-order differential equation, by using $g(\eta) = f'(\eta)$. This gives

$$-g \left(\frac{\eta}{2} - \frac{1}{\eta} \right) = g' = \frac{dg}{d\eta}.$$

By separation of variables,

$$\frac{dg}{g} = \left(\frac{1}{\eta} - \frac{\eta}{2} \right) d\eta.$$

Integrating both sides, we get

$$\ln g = \ln \eta - \frac{\eta^2}{4} + C,$$

where C is an arbitrary constant. Taking the exponential of both sides,

$$g = \eta e^C e^{-\eta^2/4} = \frac{df}{d\eta}.$$

Integrating again to get f , we have

$$f = -2e^C e^{-\eta^2/4} + A. \tag{10}$$

Now, we need to apply the initial and boundary conditions. By the initial condition (5), at $t = 0, r > 0 \Rightarrow \eta = \infty, f = 1$. Plugged into (10), this gives us

$$f(\infty) = -2e^C e^{-(\infty)^2/4} + A = 1, \text{ so } A = 1.$$

The boundary condition (6) requires $f = 0$ for $r = 0$ and $t > 0$, i.e. $f(\eta = 0) = 0$. This gives

$$f(0) = -2e^C + 1 = 0, \text{ i.e. } e^C = \frac{1}{2}.$$

With these conditions, our solution for the circulation, Γ , is

$$\Gamma = \Gamma_0 \left(1 - e^{-r^2/4\nu t} \right),$$

giving the velocity field as

$$u_\theta = \frac{\Gamma_0}{2\pi r} \left(1 - e^{-r^2/4\nu t} \right).$$

The velocity, u_θ , at different times (different values of νt), plotted against r , is shown in Fig. 1.

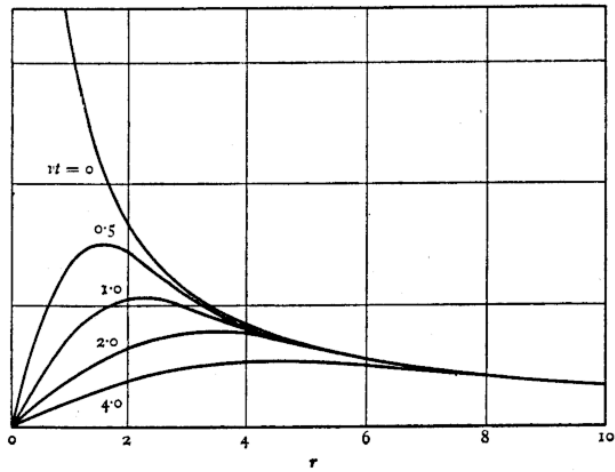


Figure 1: Plot of u_θ vs. r for $\nu t = 0, 0.5, 1, 2$ and 4 . From Batchelor, *An Introduction to Fluid Dynamics*.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 10 - D'Alembert's paradox / the velocity potential and stream function

Reading: Acheson, §4.12, 4.13, 4.1, 4.2.

We will set aside the book's chapter on waves (some of which we'll touch on later) and spend a small amount of time on the chapter about classical airfoil theory. 'Classical airfoil theory' amounts to the detailed use of ideal fluid ideas. We're interested only in taking a couple of simple results from this chapter.

1 D'Alembert's paradox

Earlier we pointed out that ideal fluid theory predicts lift well for wings until the angle of attack exceeds some threshold. There is another instructive example where ideal fluid theory fails to match with physical observations - expressed as D'Alembert's paradox, which states that *the steady, uniform flow of an ideal fluid past a fixed body imposes no drag on the body*. This can be viewed as a statement that ideal fluids are frictionless. This has obvious implications for viscous fluids....

We will need to express the notion of the momentum balance for control volumes. The book derives this in an interesting way (which gives a good lesson in the use of index notation). For an ideal fluid, we can write Euler's equation for momentum as:

$$\rho(\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla P \quad (1)$$

where we can write (labeling our Cartesian directions with the subscripts 1, 2 and 3)

$$\begin{aligned} \mathbf{u} &= (u_1, u_2, u_3) = u_i \hat{e}_i \\ \nabla &= \hat{e}_1 \frac{\partial}{\partial x_1} + \hat{e}_2 \frac{\partial}{\partial x_2} + \hat{e}_3 \frac{\partial}{\partial x_3} = \hat{e}_i \frac{\partial}{\partial x_i} \\ \mathbf{u} \cdot \nabla &= u_1 \frac{\partial}{\partial x_1} + u_2 \frac{\partial}{\partial x_2} + u_3 \frac{\partial}{\partial x_3} = u_i \frac{\partial}{\partial x_i} \end{aligned}$$

making use of the implied summation convention for repeated indices. We can then write (1) as

$$\rho u_j \frac{\partial}{\partial x_j} u_i = -\frac{\partial p}{\partial x_i} \quad (2)$$

which is strictly speaking the equation for the x_i -component of (1).

Now we want to integrate this over a fixed volume V enclosed by a surface S with local outward normal vector $\hat{\mathbf{n}}$. For the left hand side, we will use the following:

$$\frac{\partial}{\partial x_j} (u_j u_i) = u_j \frac{\partial u_i}{\partial x_j} + u_i \frac{\partial u_j}{\partial x_j}.$$

But, by the implied summation convention, the second term on the right hand side is

$$u_i \frac{\partial u_j}{\partial x_j} = u_i \left(\frac{\partial u_1}{\partial x_1} + \frac{\partial u_2}{\partial x_2} + \frac{\partial u_3}{\partial x_3} \right) = u_i (\nabla \cdot \mathbf{u}) = 0$$

because $\nabla \cdot \mathbf{u}$ for ideal fluids. So we end up with

$$\frac{\partial}{\partial x_j} (u_j u_i) = u_j \frac{\partial u_i}{\partial x_j}. \quad (3)$$

Now we want to integrate (3) over V , i.e. we want to find (using $\rho=\text{const.}$)

$$\rho \int_V \frac{\partial}{\partial x_j} (u_j u_i). \quad (4)$$

But observe that we can write

$$\frac{\partial}{\partial x_j} (u_j u_i) = \nabla \cdot (u_i \mathbf{u})$$

(think about this), and then we have for (4)

$$\rho \int_V \frac{\partial}{\partial x_j} (u_j u_i) = \rho \int_V \nabla \cdot (u_i \mathbf{u}) dV = \rho \int_S (u_i \mathbf{u}) \cdot \hat{\mathbf{n}} dS = \rho \int_S u_i u_j n_j dS \quad (5)$$

where we have applied the implied summation to the $\mathbf{u} \cdot \hat{\mathbf{n}}$ dot product.

For the right hand side of (2), our integral is

$$- \int_V \frac{\partial p}{\partial x_i} dV = - \int_S p n_i dS = - \int_S p \hat{\mathbf{n}} dS$$

(using an identity in the text derived from the divergence theorem). The result of integrating (2) is thus

$$- \int_S p \hat{\mathbf{n}} dS = \rho \int_S \mathbf{u} (\mathbf{u} \cdot \hat{\mathbf{n}}) dS$$

which is exactly what you'd find from a control volume analysis.

2 The velocity potential and streamfunction

Section 4.2 in the text goes over the justification for the use of the velocity potential for the case of *irrotational* flow (i.e. vorticity, $\omega = \nabla \times \mathbf{u}$, = 0)

$$\mathbf{u} = \nabla \phi, \quad \text{where } \phi \text{ is a scalar function}$$

in more detail than we did in the first homework. Note especially that in going from Eq. 4.2 to 4.3, Acheson makes use of Appendix A.2.

The streamfunction is another useful formulation for the velocity field, and is valid for *incompressible, two-dimensional* flow, where $\nabla \cdot \mathbf{u} = 0$. The fundamental idea is to use the vector identity that

$$\nabla \cdot (\nabla \times \mathbf{f}) = 0 \quad \text{for any vector } f$$

i.e. the divergence of the curl of any vector is zero. This justifies Eq. 4.8 in the text.

Important note: the velocity potential and the streamfunction are *not* limited to ideal fluids.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 11 - Vortex motion: induced velocity, Kelvin's circulation theorem

Reading: Acheson, §5.1-5.3.

It is sometimes convenient to formulate flow problems in terms of the vorticity rather than the velocity. Not only are a lot of interesting flow systems characterized by rotation, but the equations for the time evolution of the vorticity can be simpler because they don't involve pressure. If you're doing computer simulations, for example, not having to compute the pressure might simplify your task considerably.

1 Velocity field induced by a vortex

Of course, computing a flow field in terms of the vorticity makes little sense if we can't get back the velocity from the vorticity field. The way we do this is through the *Biot-Savart law*, which tells us the velocity field induced at a point \mathbf{x} by a given vorticity field, as

$$\mathbf{u}(\mathbf{x}) = \frac{1}{4\pi} \int \frac{\boldsymbol{\omega}(\mathbf{x}') \times \mathbf{s}}{s^3} dV. \quad (1)$$

In this equation, \mathbf{x} is the point at which we're computing the velocity field, the integrand tells us the velocity induced at \mathbf{x} by the vorticity in a volume element dV located at \mathbf{x}' , \mathbf{s} is the vector (with magnitude s) connecting the point \mathbf{x}' to \mathbf{x} , and the integral is taken over all space (specifically, all space in which the vorticity is non-zero). The process by which a vorticity field induces a velocity field is exactly analogous to that by which an electric current induces a magnetic field.

In the form of (1), the Biot-Savart law is written for vorticity fields distributed over volumes, since the integrand includes the volume element dV . We are also interested in describing vorticity fields that are concentrated into lines (in the electromagnetic analogy, this corresponds to line currents). For such vorticity fields, the Biot-Savart law becomes

$$\mathbf{u}(\mathbf{x}) = \frac{1}{4\pi} \int \frac{\kappa d\mathbf{l}(\mathbf{x}') \times \mathbf{s}}{s^3} \quad (2)$$

where $d\mathbf{l}(\mathbf{x}')$ is a length element along a vortex line located at \mathbf{x}' , and the integral is taken over all vortex lines that might be present. The vortex line strength is denoted by κ , *not* by the vorticity, $\boldsymbol{\omega}$, as appeared in (1). There are two immediate explanations for this. The first is that the vorticity is undefined on vortex lines, as we recall from Chapter 2. The second is that the vorticity has the wrong units for (2).

We will illustrate the use of (2) by computing the velocity field induced by a straight, infinite vortex line of uniform strength κ aligned with the positive z -axis (Fig. 1). Consider a point located a distance r from the vortex line. Without loss of generality, we can assume that the point lies at $z = 0$. The line element $d\mathbf{l}$ points in the positive z -direction, so

$$d\mathbf{l} = dz \hat{e}_z.$$

The cross-product $d\mathbf{l} \times \mathbf{s}$ therefore works out as

$$d\mathbf{l} \times \mathbf{s} = dz s \sin \psi \hat{e}_\theta,$$

where \hat{e}_θ points along the dotted loop in the figure in the sense indicated, by the right-hand rule. The magnitude s is given by

$$s = \sqrt{z^2 + r^2},$$

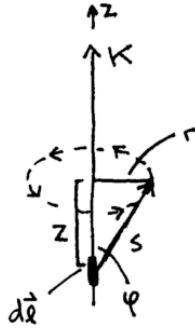


Figure 1: Velocity induced by a vortex line.

since the line element $d\mathbf{l}$ in question is located at z , so $\sin \psi$ can be written

$$\sin \psi = \frac{r}{s} = \frac{r}{\sqrt{z^2 + r^2}}.$$

The relation (2) thus becomes

$$\begin{aligned} \mathbf{u}(r) &= \frac{\kappa \hat{e}_\theta}{4\pi} \int_{z=-\infty}^{\infty} \frac{r}{[r^2 + z^2]^{3/2}} dz \\ &= \frac{\kappa \hat{e}_\theta}{4\pi r^2} \int_{z=-\infty}^{\infty} \frac{1}{[1 + (z/r)^2]^{3/2}} dz \end{aligned} \quad (3)$$

where we can take the r out of the integral since the integral is over z . To solve the integral, we make the substitution

$$z/r = \tan \alpha$$

and use the following relations:

$$\begin{aligned} dz &= r \sec^2 \alpha d\alpha \\ 1 + \tan^2 \alpha &= \sec^2 \alpha \\ z = \pm\infty &\Rightarrow \alpha = \pm\frac{\pi}{2}. \end{aligned}$$

Equation (3) thus becomes

$$\mathbf{u}(r) = \frac{\kappa \hat{e}_\theta}{4\pi r} \int_{\alpha=-\pi/2}^{\pi/2} \frac{1}{\sec \alpha} d\alpha = \frac{\kappa \hat{e}_\theta}{4\pi r} \int_{\alpha=-\pi/2}^{\pi/2} \cos \alpha d\alpha = \frac{\kappa \hat{e}_\theta}{4\pi r} [\sin \alpha]_{-\pi/2}^{\pi/2} = \frac{\kappa}{2\pi r} \hat{e}_\theta. \quad (4)$$

In Chapter 2 of the text (p.46) we were given the velocity field of a line vortex as

$$\mathbf{u} = \frac{\Gamma_0}{2\pi r} \hat{e}_\theta. \quad (5)$$

By comparison of (4) and (5), then, we see that the measure of the strength of a line vortex, which we called κ in (2), is in fact the circulation Γ_0 induced by the vortex.

2 Theorems of vortex motion: Kelvin's circulation theorem

Kelvin's circulation theorem is expressed as follows: given an ideal fluid subject to conservative body forces (e.g. gravity), and with $C(t)$ being a closed circuit consisting of the same fluid particles as time proceeds, the circulation

$$\Gamma = \int_{C(t)} \mathbf{u} \cdot d\mathbf{x}$$

is not a function of time. The proof of this is outlined in the text (in §5.1, and also in Exercise 5.2, which is explained in the back of the book).

A key point is that there is no condition in the theorem requiring the surface enclosed by $C(t)$ to lie entirely in the fluid. In particular, it is permissible for $C(t)$ to enclose a wing, for example. This completes the explanation of the generation of lift on a wing started in Chapter 1 (explained in §5.1).

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 11a - Proof of Kelvin's circulation theorem

Kelvin's circulation theorem. Consider an ideal fluid moving in the presence of a conservative body force (e.g. gravity). Let $C(t)$ be a closed circuit consisting of the same fluid particles as time proceeds. Then the circulation

$$\Gamma = \int_{C(t)} \mathbf{u} \cdot d\mathbf{x}$$

is independent of time.

Proof. We will need first to prove the following relation:

$$\frac{d}{dt} \int_{C(t)} \mathbf{u} \cdot d\mathbf{x} = \int_{C(t)} \frac{D\mathbf{u}}{Dt} \cdot d\mathbf{x}.$$

We start by representing the points on the curve $C(t)$ by $\mathbf{x}(s, t)$, where $s \in [0, 1]$. That is, we are parameterizing C with the variable s , so that every fluid particle on the curve C gets assigned a unique value of s between 0 and 1. An example of how this parameterization works is given on the left in Fig. 1. To express $d\mathbf{x}$, the vector line element along the curve C , in terms of

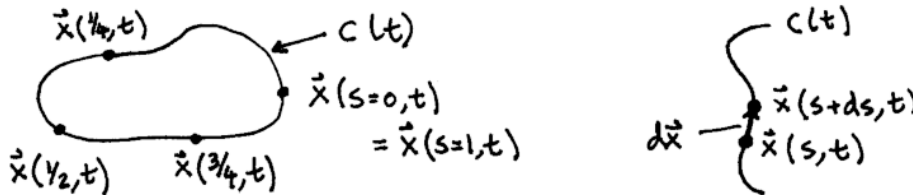


Figure 1: Sample parameterization of a curve C in terms of s (left); determination of a line element $d\mathbf{x}$ along C (right).

this parameterization, look at the other cartoon in Fig. 1. Consider two points on C , $\mathbf{x}(s, t)$ and $\mathbf{x}(s+ds, t)$. Let $d\mathbf{x}$ be the vector that goes from $\mathbf{x}(s, t)$ to $\mathbf{x}(s+ds, t)$, i.e. $\mathbf{x}(s, t) + d\mathbf{x} = \mathbf{x}(s+ds, t)$. Thus,

$$d\mathbf{x} = \mathbf{x}(s+ds, t) - \mathbf{x}(s, t).$$

Clearly, as ds becomes small, then $d\mathbf{x}$ becomes tangent to the curve C , and the above relation becomes

$$d\mathbf{x} = \left[\frac{\mathbf{x}(s+ds, t) - \mathbf{x}(s, t)}{ds} \right] ds = \frac{\partial \mathbf{x}}{\partial s} ds \quad \text{as } ds \rightarrow 0 \quad (1)$$

where the quotient is the partial derivative $\frac{\partial \mathbf{x}}{\partial s}$ because t is held constant.

This expression for $d\mathbf{x}$ allows us to write

$$\frac{d}{dt} \int_{C(t)} \mathbf{u} \cdot d\mathbf{x} = \frac{d}{dt} \int_{s=0}^1 \mathbf{u} \cdot \frac{\partial \mathbf{x}}{\partial s} ds. \quad (2)$$

The limits $s = 0$ and 1 on the second integral reflect the parameterization, whereby going from $s = 0$ to 1 takes us once around the curve $C(t)$. Since s and t are independent variables in the

parametric representation of C , we can put the time derivative d/dt inside the s integral as long as we write it as the partial derivative $\partial/\partial t$, with s held constant. Thus

$$\frac{d}{dt} \int_{s=0}^1 \mathbf{u} \cdot \frac{\partial \mathbf{x}}{\partial s} ds = \int_0^1 \frac{\partial}{\partial t} \left(\mathbf{u} \cdot \frac{\partial \mathbf{x}}{\partial s} \right) ds. \quad (3)$$

By the product rule, we have for the term in the second integrand

$$\frac{\partial}{\partial t} \left(\mathbf{u} \cdot \frac{\partial \mathbf{x}}{\partial s} \right) = \left(\frac{\partial \mathbf{u}}{\partial t} \right)_s \cdot \left(\frac{\partial \mathbf{x}}{\partial s} \right)_t + \mathbf{u} \cdot \left(\frac{\partial}{\partial t} \right)_s \left(\frac{\partial \mathbf{x}}{\partial s} \right)_t.$$

The term $(\partial \mathbf{u} / \partial t)_s$ is the time derivative of the velocity for a particular fluid element (fixed value of s), which is by definition the same as the material derivative $D\mathbf{u}/Dt$. For the second term on the right hand side, we can interchange the order of partial differentiation, so

$$\mathbf{u} \cdot \left(\frac{\partial}{\partial t} \right)_s \left(\frac{\partial \mathbf{x}}{\partial s} \right)_t = \mathbf{u} \cdot \left(\frac{\partial}{\partial s} \right)_t \left(\frac{\partial \mathbf{x}}{\partial t} \right)_s = \mathbf{u} \cdot \left(\frac{\partial \mathbf{u}}{\partial s} \right)_t = \frac{\partial}{\partial s} \left(\frac{1}{2} \mathbf{u} \cdot \mathbf{u} \right).$$

This allows us to rewrite (3) as

$$\int_0^1 \frac{\partial}{\partial t} \left(\mathbf{u} \cdot \frac{\partial \mathbf{x}}{\partial s} \right) ds = \int_0^1 \left[\frac{D\mathbf{u}}{Dt} \cdot \frac{\partial \mathbf{x}}{\partial s} + \frac{\partial}{\partial s} \left(\frac{1}{2} \mathbf{u} \cdot \mathbf{u} \right) \right] ds. \quad (4)$$

Note that the second term in the integrand just evaluates as

$$\int_0^1 \frac{\partial}{\partial s} \left(\frac{1}{2} \mathbf{u} \cdot \mathbf{u} \right) ds = \left[\frac{1}{2} \mathbf{u} \cdot \mathbf{u} \right]_0^1 = 0,$$

because $s = 0$ and $s = 1$ represent the same point in space, and \mathbf{u} is single-valued in space (meaning it has only one value at any point). So, combining this result with (4), (3) and (2), we have

$$\frac{d}{dt} \int_{C(t)} \mathbf{u} \cdot d\mathbf{x} = \int_0^1 \frac{D\mathbf{u}}{Dt} \cdot \frac{\partial \mathbf{x}}{\partial s} ds = \int_{C(t)} \frac{D\mathbf{u}}{Dt} \cdot d\mathbf{x}, \quad (5)$$

where we have used (1) in the last step.

Now we go back to Euler's equation for the velocity field (since we are dealing with an ideal fluid)

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho} \nabla p + \mathbf{g} = -\nabla \left(\frac{p}{\rho} + \chi \right)$$

where we have written $\mathbf{g} = -\nabla \chi$ since gravity is conservative, and we have moved ρ inside the derivative because it's a constant for ideal fluids. Then, (5) becomes

$$\frac{d\Gamma}{dt} = - \int_{C(t)} \nabla \left(\frac{p}{\rho} + \chi \right) \cdot d\mathbf{x}. \quad (6)$$

To interpret this integral, recall some of the properties of gradients. Let ϕ be a scalar function. If $\hat{\mathbf{r}}$ is a unit vector, then we had

$$\nabla \phi \cdot \hat{\mathbf{r}} = \frac{d\phi}{dr},$$

which means that the gradient of ϕ dotted into the unit vector $\hat{\mathbf{r}}$ is equal to the derivative of ϕ in the $\hat{\mathbf{r}}$ direction. Now consider the vector line element $d\mathbf{x}$, which we will write as $d\mathbf{x} = dx\hat{\mathbf{x}}$. So, dx is the magnitude of $d\mathbf{x}$, and $\hat{\mathbf{x}}$ is its direction. We then have

$$\nabla\phi \cdot d\mathbf{x} = (\nabla\phi \cdot \hat{\mathbf{x}})dx = \frac{d\phi}{dx}dx = d\phi,$$

where this $d\phi$ is the change in ϕ on moving a distance dx in the $\hat{\mathbf{x}}$ direction.

Now we look again at (6). Using the results for the gradient above, we can write

$$-\int_{C(t)} \nabla \left(\frac{p}{\rho} + \chi \right) \cdot d\mathbf{x} = -\int_{C(t)} d \left[\frac{p}{\rho} + \chi \right] = - \left[\overbrace{\left(\frac{p}{\rho} + \chi \right)}^{\text{at end of curve } C(t)} - \overbrace{\left(\frac{p}{\rho} + \chi \right)}^{\text{at start of curve } C(t)} \right] = 0,$$

where the result is zero because $C(t)$ is a closed curve (so its start and end are the same point), and p , ρ and χ are single-valued functions. Thus

$$\frac{d\Gamma}{dt} = 0,$$

so the circulation Γ is not a function of time. Q.E.D.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 12 - Vortex motion: vortex lines, Helmholtz's vortex theorems

Reading: Acheson, §5.2, 5.3.

A *vortex line* is a curve which is parallel to the instantaneous vorticity vector $\omega = \nabla \times \mathbf{u}$ at all of its points. A vortex line thus relates to the vorticity in the same way that a streamline relates to the velocity. So, if we define a vortex line as $\mathbf{x} = (x(s), y(s), z(s))$, and the vorticity as $\omega = (\omega_x, \omega_y, \omega_z)$, then the following relations hold:

$$\frac{dx/ds}{\omega_x} = \frac{dy/ds}{\omega_y} = \frac{dz/ds}{\omega_z}$$

at all points on a vortex line. The slope of a vortex line, as projected into the $x - y$ plane, is then $dy/dx = \omega_y/\omega_x$, with similar results for the $y - z$ and $x - z$ planes.

We introduced the concept of *vortex tubes* in the second homework assignment. The book defines a vortex tube as the set of vortex lines that pass through a simple closed curve in space. (By simple closed curve we basically mean that the curve is connected – i.e. one piece – and doesn't cross itself.)

One Helmholtz vortex theorem deals with vortex lines; the other, with vortex tubes. Both are formulated for ideal fluids in the presence of conservative body forces (like gravity), which are the same conditions for which we formulated Kelvin's circulation theorem.

1 The Helmholtz vortex theorems

Helmholtz's first vortex theorem. *The fluid elements that lie on a vortex line at some instant continue to lie on a vortex line, that is, vortex lines 'move with the fluid.'*

To visualize this, suppose we mark a vortex line at some time by, say, coloring the specific fluid particles that make up the line. Then at all later times, the marked particles will continue to constitute a vortex line. By this theorem, we can also say that vortex *tubes* move with the fluid.

The proof in the book proceeds by defining a vortex surface as a surface that is everywhere tangent to the vorticity vector ω . By applying Kelvin's circulation theorem and Stokes's theorem, you can show that the fluid particles that make up the vortex surface at any point in time will continue to make up a vortex surface for all times, that is, the vorticity will always be tangent to that set of fluid particles. Now, pick one vortex line out of that vortex surface, and construct another, separate surface that's tangent to the vorticity everywhere and includes that vortex line. Then the vorticity will always be tangent to *those* fluid particles also. The intersection of the two different surfaces thus remains a vortex line. Because the two surfaces are defined in terms of fluid particles and thus move with the flow, this means that the vortex line defined by the intersection of the surfaces moves with the flow.

Helmholtz's second vortex theorem. *The quantity*

$$\Gamma = \int_S \omega \cdot \hat{\mathbf{n}} dS$$

is the same for all cross-sectional surfaces S cut through a vortex tube. Also, Γ is not a function of time.

The proof of this theorem is very similar to the argument we applied in the first homework assignment to show that vortex tubes can't end in a fluid. The added step is to equate the vorticity 'flux' through a cross-section of the tube ($\int_S \omega \cdot \hat{\mathbf{n}} dS$) to the circulation around the cross-section using Stokes's theorem. Because vortex lines move with the fluid (by the above theorem), any curve enclosing a vortex tube also moves with the fluid. Then by Kelvin's circulation theorem, the circulation Γ around such a curve remains constant.

The interesting interpretation of this theorem is in terms of thin vortex tubes where the vorticity is essentially constant across any cross-section. In that case, the theorem tells us that if the cross-sectional area decreases, the vorticity has to increase. Since the fluid is incompressible, the cross-sectional area decreases if we stretch a particular set of fluid elements lengthwise (recall that vortex tubes consist of a fixed set of fluid elements, from the first theorem). Thus, stretching a vortex tube lengthwise results in increased vorticity. This has interesting interpretations in analogies with other physical examples - think especially of the conservation of angular momentum.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 13 - Vortex motion: interpretation of Helmholtz's vortex theorems

Reading: Acheson, §5.3-5.5.

In the last lecture we covered Helmholtz's (two) vortex theorems, which apply to ideal fluids subject to conservative body forces. The theorems state:

- The fluid elements that lie on a vortex line at some point in time continue to define a vortex line. This means that vortex lines move with the fluid.
- Given a vortex tube, the circulation,

$$\Gamma = \int_S \boldsymbol{\omega} \cdot \hat{\mathbf{n}} dS, \quad (\text{using Stokes's theorem})$$

is the same for all cross-sections S cut through the vortex tube, and is the same for all times.

The second of these two theorems in particular lends itself to ready interpretation in terms of physical intuition.

1 Thin vortex tubes

If a vortex tube is thin enough, then the vorticity will be virtually constant across the tube's cross-section. We will consider the special case of a thin, cylindrical vortex tube, defined as the set of vortex lines that pass through a small circle drawn in the flow, with the additional assumption that the vortex lines are parallel (Fig. 1) Consider a finite segment cut from such a vortex tube, with

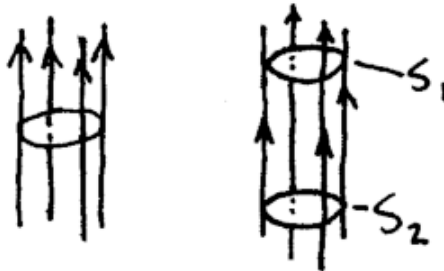


Figure 1: A thin, cylindrical vortex tube.

length l . Label the circular cross-sections that form the ends of this finite segment as S_1 and S_2 , and denote the area of these faces as δS . We will focus our attention on face S_1 . Because its area, δS , is small, we will assume that the vorticity on any point on that face has the constant value ω_1 . Then the circulation around the circle C_1 that encloses the surface S_1 is

$$\Gamma_1 = \int_{C_1} \mathbf{u} \cdot d\mathbf{x} = (\text{Stokes's theorem}) \int_{S_1} \boldsymbol{\omega} \cdot \hat{\mathbf{n}} dS = \omega \delta S. \quad (1)$$

Now consider what happens if we change the length of the finite segment of the vortex tube.

Specifically, we will consider the set of fluid elements initially contained in the segment of length l cut from the vortex tube. Because ideal fluids are incompressible, the volume occupied by that set of fluid elements has to be constant. Assume that the segment we're considering remains cylindrical even as its length changes. The volume of the fluid element is $l\delta S$, so as depicted in Fig. 2, if we

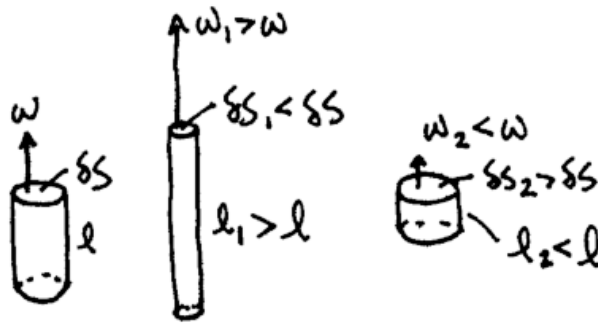


Figure 2: Stretching/contracting a vortex tube.

change its length from l to $l_1 > l$, then the cross-sectional area has to decrease (proportionally), giving $\delta S_1 < \delta S$. Similarly, if $l_2 < l$, then $\delta S_2 > \delta S$ (proportionally). But by (1), $\omega \delta S$ is constant, so:

- If we stretch a thin segment of vortex tube ($l_1 > l$), its vorticity increases (proportionally), giving $\omega_1 > \omega$.
- If we contract a thin segment of vortex tube ($l_2 < l$), its vorticity decreases (proportionally), $\omega_2 < \omega$.

1.1 Tornadoes



Figure 3: Tornado formation.

This vortex stretching process is what happens in a type of tornado (the non-supercell variety). Initially, suppose there's some small vorticity near the ground, distributed over a wide area, perhaps in the form of a large, slow circulation caused by the presence of hills or mountains or something. If that slow rotation meets with strong updrafts, maybe due to thermal convection, then the effect is to stretch the vortex tube defined by that circulation. As the tube gets stretched to the characteristic thin tornado profile, the vorticity increases, accounting for the strong rotation of the tornado. As Acheson points out, Helmholtz's first theorem is then illustrated as the tornado subsequently moves to follow the pattern of clouds overhead.

1.2 Relation to the vorticity equation

We can also get at Helmholtz's second theorem by looking at the Euler vorticity equation:

$$\frac{D\omega}{Dt} = (\omega \cdot \nabla)\mathbf{u}.$$

Suppose that the vortex lines are almost perfectly aligned with the z -axis, so

$$\boldsymbol{\omega} \approx (0, 0, \omega_z). \quad (2)$$

In that case,

$$\boldsymbol{\omega} \cdot \nabla \approx \omega_z \frac{\partial}{\partial z}$$

and we can thus write for the z -component of the equation

$$\frac{D\omega_z}{Dt} \approx \omega_z \frac{\partial w}{\partial z} \quad (3)$$

where w is the velocity component in the z -direction. Now consider what it means for $\partial w/\partial z$ to be positive or negative. If $\partial w/\partial z$ is positive, then fluid elements located at higher z also move faster in the z -direction, i.e. there is stretching in the z -direction. If $\partial w/\partial z$ is negative, then fluid elements located at higher z are moving more slowly in the z -direction, i.e. there is contraction in the z -direction. Thus by the above expression, stretching in the z -direction increases the z -component of vorticity, since $D\omega_z/Dt > 0$, while contracting in the z -direction decreases ω_z , since $D\omega_z/Dt < 0$ in that case.

(Question: do you see a possible inconsistency between (2) and (3)?)

1.3 Angular momentum analogy

From basic physics you already know some principles that should give you some physical intuition for Helmholtz's second theorem. In particular, we know that angular momentum of a body is conserved in the absence of torques, where angular momentum is the moment of inertia multiplied by the angular velocity (which, like the vorticity, has units of $(\text{time})^{-1}$). You will remember that the moment of inertia of an object is related to its radial spread from the axis of rotation. That is, for a given mass, an object has a higher moment of inertia if it extends further from the axis of rotation. The angular velocity, by virtue of its units, describes a frequency of rotation (and thus does not relate directly to a linear velocity).

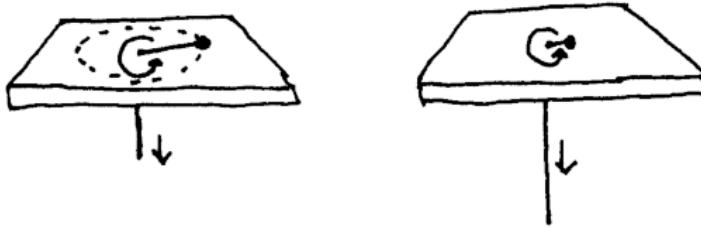


Figure 4: Effect of changing the moment of inertia.

As an illustration of this, consider Fig. 4. This is supposed to represent a table with a small hole cut in the center. A small weight attached to a string sits on the table, and the other end of the string reaches through the hole. Suppose we set the weight into motion so it swings around at the end of the string. If we pull on the string from below the table, we decrease the moment of inertia of the weight by decreasing its radial distance from the axis of rotation. Then, by conservation of angular momentum, the frequency of its rotation around the hole in the table must increase.

This essentially describes what happens when you stretch or contract a segment vortex tube. The mass of a set of fluid elements that make up the segment is constant, but by stretching you decrease the radius and thus decrease the moment of inertia of the segment. It compensates by increasing its rotation rate (i.e. its vorticity). This is the gist of Helmholtz's second theorem.

1.4 Something to think about

How does this relate to the vortex formed by draining a tub? What's the approximate angular velocity of the tub vortex? If we treat the water in the tub as ideal, where would the vorticity in the tub vortex come from (look at §5.2 in the book)? What about the venerable explanation that the Coriolis force causes tub vortices to rotate in different directions in the Northern and Southern Hemispheres? The Coriolis force is linearly related to the Earth's rotation rate. How does this compare with the tub vortex's rotation rate? Is it reasonable for the Coriolis force to affect the rotation in the tub vortex?

1.5 A cautionary note

In discussing the Helmholtz theorems we have made a number of simplifying assumptions. The theorems themselves assume fluids are ideal; and we have also made things easier on ourselves by talking about 'thin' vortex tubes, that are aligned with the z -axis, etc. These conditions won't necessarily apply with viscous fluids, but the qualitative conclusions are pretty general. Just remember what assumptions we've made and why they might fail. An example: what fluid property might make it problematic to apply conservation of angular momentum, and why?

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 14 - Vortex motion: axisymmetric flow, spherical coordinates

Reading: Acheson, §5.3-5.5.

We now want to look at some problems that represent *axisymmetric flow*. This is analogous to two-dimensional flow in Cartesian coordinates, in that the flow has non-zero velocity components in two directions, and these components are only allowed to vary in those two directions. The difference is that with axisymmetric flow, the coordinates of interest are referred to a single axis or pole – for example, cylindrical coordinates – and the coordinates that are allowed to vary are the coordinate that measures distance along the polar axis and the coordinate that measures distance away from the polar axis. Mathematically, the velocity field for axisymmetric flow is expressed in cylindrical coordinates as:

$$\mathbf{u} = (u_r(r, z, t), 0, u_z(r, z, t)). \quad (1)$$

Because $u_\theta = 0$, the streamlines of an axisymmetric flow will be confined to any plane that includes the z -axis, and because \mathbf{u} is not a function of θ , the streamlines are identical for all such planes (Fig. 1). This explains the nomenclature ‘axisymmetric.’

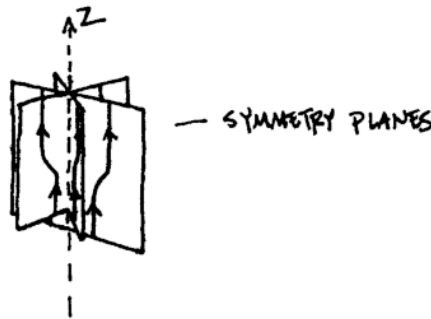


Figure 1: Symmetry planes for axisymmetric flow.

The vorticity field for an axisymmetric flow is given by

$$\begin{aligned} \boldsymbol{\omega} = \nabla \times \mathbf{u} &= \frac{1}{r} \begin{vmatrix} \hat{e}_r & r\hat{e}_\theta & \hat{e}_z \\ \frac{\partial}{\partial r} & \frac{\partial}{\partial \theta} & \frac{\partial}{\partial z} \\ u_r & ru_\theta & r_z \end{vmatrix} \\ &= \frac{1}{r} \begin{vmatrix} \hat{e}_r & r\hat{e}_\theta & \hat{e}_z \\ \frac{\partial}{\partial r} & 0 & \frac{\partial}{\partial z} \\ u_r & 0 & r_z \end{vmatrix} \\ &= \left[\frac{\partial u_r}{\partial z} - \frac{\partial u_z}{\partial r} \right] \hat{e}_\theta. \end{aligned}$$

So, for axisymmetric flow, the vorticity vectors are aligned only in the θ -direction, which means that vortex lines are circular and centered on the z -axis. Vortex rings, commonly seen in the form of smoke rings, represent an axisymmetric flow where the vorticity is concentrated in a thin tube. (This is treated in detail in the text.)

1 Spherical coordinates

Axisymmetric flow can also, and sometimes more conveniently, be described in *spherical coordinates*. As shown in Fig. 2, for an arbitrary point in space, the spherical coordinate r measures the distance of the point from the origin, the coordinate θ measures the angle between the Cartesian z -axis and the \mathbf{r} vector, and the coordinate ϕ is the angle between the x -axis and the projection of \mathbf{r} into the $x - y$ plane. Remember that the cylindrical coordinate system (r, θ, z) is so named because if we

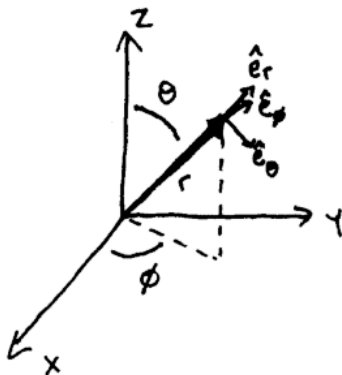


Figure 2: The spherical coordinate system.

let the coordinate r vary from 0 to R , let z vary from 0 to h , and let θ vary over the full range 0 to 2π , we describe a cylinder of height h and cross-sectional radius R . Similarly, for the spherical coordinate system (r, ϕ, θ) , if we let r vary from 0 to R , let ϕ vary over its full range 0 to 2π , and let θ vary over *its* full range 0 to π (why not 2π ?), we describe a sphere of radius R .

It can be seen that the spherical coordinate ϕ is identical to the cylindrical coordinate θ , which means that the cylindrical coordinates r and z can be transformed into the spherical coordinates r and θ . This means, then, that the cylindrical coordinate description of axisymmetric flow (1) can be written in spherical coordinates as:

$$\mathbf{u} = (u_r(r, \theta, t), u_\theta(r, \theta, t), u_\phi = 0). \quad (2)$$

2 The Stokes stream function

For two-dimensional, incompressible flow in Cartesian coordinates, we learned in Chapter 4 how to write the velocity in terms of a stream function that is constant along streamlines. Since axisymmetric flow is just two-dimensional flow in cylindrical or spherical coordinates, we can reasonably expect to be able to write a stream function for axisymmetric flow too.

Recall that the stream function representation arises because for incompressible flow because for such flows,

$$\nabla \cdot \mathbf{u} = 0.$$

One of our vector identities tells us that the divergence of any curl is zero, or

$$\nabla \cdot (\nabla \times \mathbf{f}) = 0.$$

This suggests that we can write $\mathbf{u} = \nabla \times \mathbf{f}$. Such a representation is especially convenient for two-dimensional flows, because then the vector \mathbf{f} only has a component in the third dimension, the one that has no velocity component, and also the direction in which the velocity terms don't

change. For axisymmetric flow expressed in cylindrical coordinates, $\mathbf{u} = u_r \hat{e}_r + u_z \hat{e}_z$, so we can write $\mathbf{f} = \psi \hat{e}_\theta$, where ψ . Then

$$\mathbf{u} = -\frac{\partial \psi}{\partial z} \hat{e}_r + \frac{1}{r} \frac{\partial(r\psi)}{\partial r} \hat{e}_z = -\frac{\partial \psi}{\partial z} \hat{e}_r + \left(\frac{\partial \psi}{\partial r} + \frac{\psi}{r} \right) \hat{e}_z. \quad (3)$$

In Cartesian coordinates, this ψ was the stream function, so called because ψ was constant along streamlines. It turns out that ψ is *not* constant along streamlines for the axisymmetric representation (3). In particular

$$(\mathbf{u} \cdot \nabla)\psi = u_r \frac{\partial \psi}{\partial r} + u_z \frac{\partial \psi}{\partial z} = -\frac{\partial \psi}{\partial z} \frac{\partial \psi}{\partial r} + \left(\frac{\partial \psi}{\partial r} + \frac{\psi}{r} \right) \frac{\partial \psi}{\partial z} = -u_r \frac{\psi}{r}.$$

Notice, however, that if we write $\psi = \Psi/r$, so that the velocity field is written

$$\mathbf{u} = -\frac{\partial(\Psi/r)}{\partial z} \hat{e}_r + \frac{1}{r} \frac{\partial \Psi}{\partial r} \hat{e}_z = -\frac{1}{r} \frac{\partial \Psi}{\partial z} \hat{e}_r + \frac{1}{r} \frac{\partial \Psi}{\partial r} \hat{e}_z,$$

which gives

$$(\mathbf{u} \cdot \nabla)\Psi = u_r \frac{\partial \Psi}{\partial r} + u_z \frac{\partial \Psi}{\partial z} = -\frac{1}{r} \frac{\partial \Psi}{\partial z} \frac{\partial \Psi}{\partial r} + \frac{1}{r} \frac{\partial \Psi}{\partial r} \frac{\partial \Psi}{\partial z} = 0,$$

then Ψ is constant along streamlines. This Ψ is called the *Stokes stream function*.

To express these ideas in spherical coordinates, note that r in cylindrical coordinates is equal to $r \sin \theta$ in spherical coordinates, and the θ direction in cylindrical coordinates is the same as the ϕ direction in sphericals. Thus,

$$\mathbf{u} = \nabla \times (\psi \hat{e}_\phi) = \nabla \times \left(\frac{\Psi}{r \sin \theta} \hat{e}_\phi \right) = \frac{1}{r^2 \sin \theta} \frac{\partial \Psi}{\partial \theta} \hat{e}_r - \frac{1}{r \sin \theta} \frac{\partial \Psi}{\partial r} \hat{e}_\theta. \quad (4)$$

3 Irrotational flow past a sphere

Consider a sphere of radius a placed in a uniform flow (of velocity U , in the z -direction, as in Fig. 3) of ideal fluid, and realize that the flow is irrotational infinitely far upstream of the sphere. (The

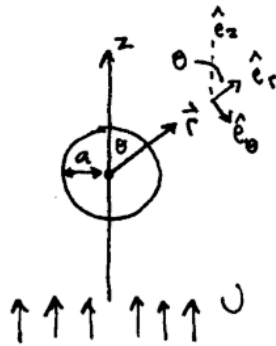


Figure 3: A sphere in uniform flow.

flow is uniform there, and a uniform flow has no gradients, so the vorticity is zero.) From §5.2, all of the flow will thus be irrotational.

For axisymmetric flow, the vorticity in spherical coordinates will only have a component in the ϕ direction, $\omega = \omega_\phi \hat{e}_\phi$, which is written

$$\omega_\phi = \frac{1}{r} \frac{\partial(r u_\theta)}{\partial r} - \frac{1}{r} \frac{\partial u_r}{\partial \theta},$$

which we can write using (4) in terms of the Stokes stream function, Ψ , as

$$\omega_\phi = -\frac{1}{r \sin \theta} \left[\frac{\partial^2 \Psi}{\partial r^2} + \frac{\sin \theta}{r^2} \frac{\partial}{\partial \theta} \left(\frac{1}{\sin \theta} \frac{\partial \Psi}{\partial \theta} \right) \right].$$

Specifying the (irrotational) flow for this problem thus comes down to solving

$$\frac{\partial^2 \Psi}{\partial r^2} + \frac{\sin \theta}{r^2} \frac{\partial}{\partial \theta} \left(\frac{1}{\sin \theta} \frac{\partial \Psi}{\partial \theta} \right) = 0 \quad (5)$$

for the region outside the sphere, i.e. $r \geq a$, subject to boundary conditions. The first condition applies at the surface:

$$\Psi = 0 \text{ on } r = a. \quad (6)$$

In particular, setting Ψ constant on the surface of the sphere ensures that the surface represents streamlines, i.e. that the velocity field at the surface is parallel to the surface. The choice of $\Psi = 0$ as opposed to any other constant is arbitrary. The other condition pertains for $r \rightarrow \infty$, or far from the sphere, where the velocity has to be uniform in the z -direction with value U . In spherical coordinates, that means (referring to Fig. 3)

$$\begin{aligned} u_r &= U \cos \theta \\ u_\theta &= -U \sin \theta. \end{aligned}$$

To express these as conditions on the stream function, Ψ , defined by (4), we have from the r -component:

$$u_r = U \cos \theta = \frac{1}{r^2 \sin \theta} \frac{\partial \Psi}{\partial \theta}$$

which gives us

$$\frac{\partial \Psi}{\partial \theta} = U r^2 \sin \theta \cos \theta$$

and, integrating over θ ,

$$\int \frac{\partial \Psi}{\partial \theta} d\theta = U r^2 \int \sin \theta \cos \theta d\theta = \frac{1}{2} U r^2 \sin^2 \theta + c_1(r). \quad (7)$$

Similarly, we have from the θ component:

$$u_\theta = -U \sin \theta = -\frac{1}{r \sin \theta} \frac{\partial \Psi}{\partial r}$$

which gives us (skipping to the integration)

$$\int \frac{\partial \Psi}{\partial r} dr = U \sin^2 \theta \int r dr = \frac{1}{2} U \sin^2 \theta r^2 + c_2(\theta). \quad (8)$$

Comparing (7) and (8), we see that c_1 and c_2 can only be a constant that isn't a function of r or θ . We can, arbitrarily, set that constant to zero since the stream function is mainly of interest in terms of its derivatives (as in (4)). Thus, we are left with

$$\Psi = \frac{1}{2} U r^2 \sin^2 \theta. \quad \text{as } r \rightarrow \infty. \quad (9)$$

This suggests a solution of the form

$$\Psi = f(r) \sin^2 \theta.$$

(Never mind how we can assume this – focus on the result.) Plugging this into (5), we get

$$f'' - 2\frac{f}{r^2} = 0,$$

which suggests that f is a sum of terms of the form r^n . We can show that these terms satisfy the differential equation only if $n = 2$ or $n = -1$. The general solution of the equation is a linear combination of these terms (again, don't sweat the details), or

$$f = Ar^2 + \frac{B}{r}$$

which, applying our boundary conditions (6) and (9), gives

$$\Psi = \frac{U}{2} \left(r^2 - \frac{a^3}{r} \right) \sin^2 \theta.$$

Applying (4), we get for the velocity components

$$\begin{aligned} u_r &= \frac{1}{r^2 \sin \theta} \frac{\partial \Psi}{\partial \theta} = U \left(1 - \frac{a^3}{r^3} \right) \cos \theta \\ u_\theta &= -\frac{1}{r \sin \theta} \frac{\partial \Psi}{\partial r} = -U \sin \theta \left(1 + \frac{a^3}{2r^3} \right). \end{aligned} \quad (10)$$

Let's look at what we have going on on the surface of the sphere. Figure 4 shows how the u_r and u_θ components look there. From (10), we have at $r = a$

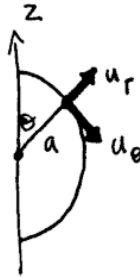


Figure 4: The velocity components on the surface of the sphere.

$$\begin{aligned} u_r(r = a) &= 0 \\ u_\theta(r = a) &= -\frac{3}{2}U \sin \theta, \end{aligned}$$

so there is no normal component of velocity on the surface, as desired, but there is a tangential component, which is allowed because the fluid is inviscid. Consider a streamline that strikes the sphere at $r = a$, $\theta \lesssim \pi$, or just off the z -axis. Initially, $\sin \theta \approx 0$, so the tangential component u_θ is small (and points in the upward direction along the surface, as shown in Fig. 5). At the equator of the sphere, $\theta = \pi/2$, u_θ is maximized, and as we move along the surface to the top of the sphere, u_θ drops sharply as $\sin \theta \rightarrow 0$. By Bernoulli's theorem (either one), this sharp drop in velocity as we move along the surface from the equator of the sphere to the top means that the pressure rises sharply along that streamline. This will become significant in terms of boundary layers, as we will see.

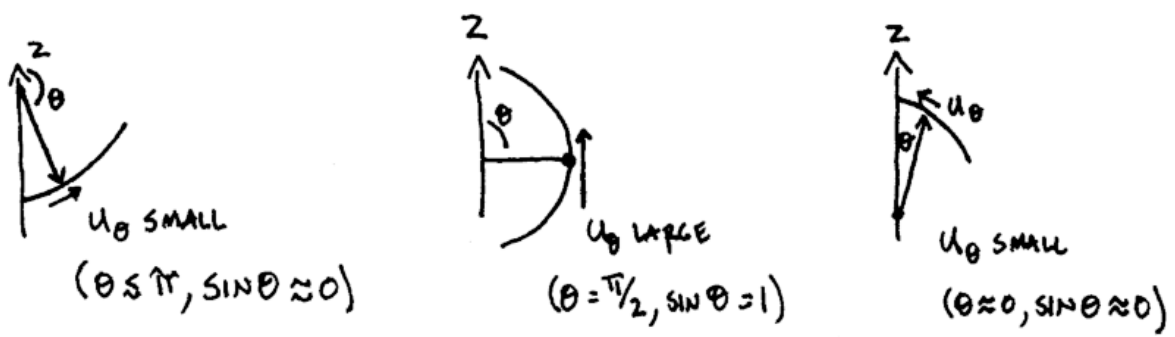


Figure 5: The velocity on a streamline at the surface of the sphere.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 15 - Vortex motion: the von Kármán vortex street, and some complex analysis

Reading: Acheson, §4.3, 5.6-5.8.

The phenomenon of the von Kármán vortex street is one of the most interesting and bizarre in fluid mechanics. And as a bonus, it has great practical significance.

The von Kármán vortex street is a particular case of the wake flow behind a circular cylinder whose long axis is perpendicular to a uniform main flow. The fascinating thing is that even if the main flow is steady (i.e. doesn't vary with time), and the cylinder is held fixed, an *unsteady* pattern of vortices will be shed behind the cylinder. The practical importance of this phenomenon is that the unsteady velocity field implies an unsteady pressure field, so that the cylinder experiences fluctuating forces. In a real flow, the cylinder in question might be a bridge pier in water, or a support cable for a television antenna, or a flagpole, where the fluctuating forces can have implications for structural stability, for example.

1 Some basic complex analysis

To treat this problem analytically we will need some basic tools from complex analysis. Define a general complex number as $z = x + iy$, where $i = \sqrt{-1}$ and x and y are real numbers. Then we refer to x as the real part of z , $x = \operatorname{Re} z$, and to y as the imaginary part of z , $y = \operatorname{Im} z$.

We choose to call the real and imaginary parts x and y because the geometric interpretation of the complex number z is as a point in a plane, where the real part of z is the horizontal axis (hence x) and the imaginary part is the vertical axis (hence y). There is also a convenient polar

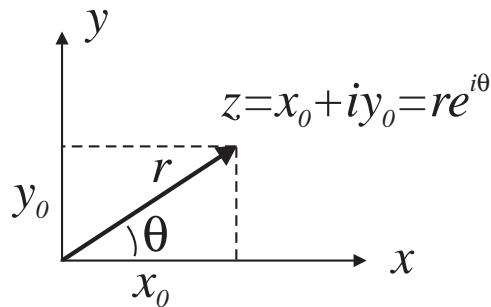


Figure 1: The complex plane.

representation of z , as seen in Fig. 1. If we denote the length of z as r , and the angle it makes with the x -axis as θ , then $x = r \cos \theta$ and $y = r \sin \theta$, so

$$z = r(\cos \theta + i \sin \theta) = re^{i\theta},$$

where we have used the expression $e^{i\theta} = \cos \theta + i \sin \theta$.

Addition and multiplication for complex numbers looks like:

$$\begin{aligned}(x_1 + iy_1) + (x_2 + iy_2) &= (x_1 + x_2) + i(y_1 + y_2) \\ (x_1 + iy_1)(x_2 + iy_2) &= (x_1x_2 - y_1y_2) + i(x_1y_2 + x_2y_1)\end{aligned}$$

as you might expect. Note that multiplication (and division) are a little cleaner if you use the polar representation $z = re^{i\theta}$.

We can, of course, have functions of the complex variable z , which are written

$$w = f(z) = g + ih,$$

with g and h being real-valued functions. We call g the real part of the function w and h the imaginary part. We can take derivatives of w exactly the same way we do for functions of real variables (well, almost exactly, but the distinctions don't really matter for us). So if $w = z^n$, for example,

$$\frac{dw}{dz} = \frac{d}{dz} z^n = n z^{n-1}.$$

There is one theorem that will be particularly useful to us. It states that *if a function $w = f(z) = g + ih$ is differentiable at some point z_0 , then*

$$\begin{aligned} \frac{\partial g}{\partial x} &= \frac{\partial h}{\partial y} \quad \text{and} \quad \frac{\partial g}{\partial y} = -\frac{\partial h}{\partial x}, \\ \text{and} \quad \frac{dw}{dz} &= \frac{\partial g}{\partial x} + i \frac{\partial h}{\partial x}. \end{aligned} \tag{1}$$

This can be verified easily for simple functions, say $w = z^2$.

2 The complex potential

Why is this interesting? Remember that we earlier said that for irrotational flow, we could write

$$\mathbf{u} = \nabla \phi$$

where ϕ was called the velocity potential. Thus, in the two-dimensional case,

$$u = \frac{\partial \phi}{\partial x}, v = \frac{\partial \phi}{\partial y} \quad (\text{for 2-D, irrotational flow}). \tag{2}$$

Also, we said that for two-dimensional, incompressible flow, we could write

$$u = \frac{\partial \psi}{\partial y}, v = -\frac{\partial \psi}{\partial x} \quad (\text{for 2-D, incompressible flow}), \tag{3}$$

where ψ was called the stream function. Now observe that (2) and (3) together look like (1), if we let the velocity potential ϕ be the function g and the stream function ψ be h . For *irrotational, incompressible two-dimensional flow*, we can then write the *complex potential*, w , as

$$w(z) = \phi + i\psi, \tag{4}$$

where the complex variable $z = x + iy$, using the real spatial variables x and y . By (1), and using (2) and (3), we then have

$$\frac{dw}{dz} = \frac{\partial \phi}{\partial x} + i \frac{\partial \psi}{\partial x} = u - iv, \tag{5}$$

which tells us how to get the velocity components back from the complex potential in Cartesian coordinates. If we're using polar coordinates, then

$$\begin{aligned} u_r &= \frac{\partial \phi}{\partial r} = \frac{1}{r} \frac{\partial \psi}{\partial \theta} \\ u_\theta &= \frac{1}{r} \frac{\partial \phi}{\partial \theta} = -\frac{\partial \psi}{\partial r}. \end{aligned} \tag{6}$$

The text works through a couple of simple examples of the use of the complex potential. The one we'll use is the complex potential representation for the velocity field of a line vortex. For a line vortex located at the origin,

$$\mathbf{u} = \frac{\Gamma}{2\pi r} \hat{e}_\theta.$$

The complex potential for this is

$$w = -\frac{i\Gamma}{2\pi} \ln(z).$$

If the line vortex is located at some point $z = z_0 = x_0 + iy_0$, then the complex potential is

$$w = -\frac{i\Gamma}{2\pi} \ln(z - z_0). \quad (7)$$

3 The von Kármán vortex street

The text has a good discussion of the development of the von Kármán vortex street which I won't bother to duplicate. Then, Acheson models the vortex street (for a left-to-right mean flow) mathematically as an infinite set of line vortices, arranged in two rows. In the lower row, the vortices are

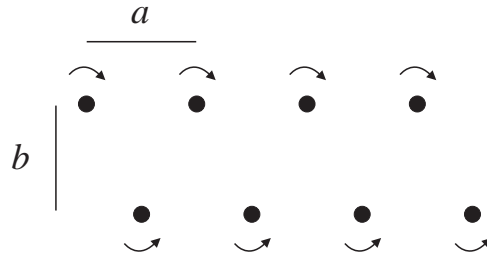


Figure 2: A model of the von Kármán vortex street. The mean flow is from left to right.

at $z = na$, where n is any integer, and have strength Γ , corresponding to a counterclockwise rotation. In the upper row, the vortices are at $z = (n + 1/2)a + ib$ and have strength $-\Gamma$, corresponding to a clockwise rotation.

There are a couple of useful geometric observations. First, we point out that *the velocity of any single line vortex will be that induced by every vortex other than itself*. Suppose we consider a vortex on either row. By symmetry, the induced velocities of the vortices in the same row cancel in pairs, i.e. the induced velocity of the vortex located a distance ma away cancels with that located a distance $-ma$ away. Similarly, the y -components of the induced velocities of the vortices in the other row cancel symmetrically, but the x -components of the induced velocities add up to give a leftward velocity. Thus, the vortex street moves to the left under the action of its own vorticity.

The text works through the math of determining the velocity of one of the vortices by adding the respective complex potentials together for the vortices in the other row. For a vortex on the top row, the complex potential of all of the vortices on the bottom row is

$$w = -\frac{i\Gamma}{2\pi} \ln \left(\sin \frac{\pi z}{a} \right) + \text{const.}$$

The resulting speed of the vortex street is

$$V = \frac{\Gamma}{2a} \tanh \left(\frac{\pi b}{a} \right).$$

In the frame of reference where the cylinder is moving in a fixed stream, the vortex street follows the cylinder but gradually falls behind.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 16 - Vortex motion: effects of viscosity

Reading: Acheson, §5.9, 5.10.

In Chapter 2 we looked at the vorticity equation for viscous fluids, which was

$$\frac{\partial \omega}{\partial t} + (\mathbf{u} \cdot \nabla) \omega = (\omega \cdot \nabla) \mathbf{u} + \nu \nabla^2 \omega. \quad (1)$$

The second term on the left side of (1) is clearly recognizable as the advective term in the material time derivative, and describes the change in the vorticity of a fluid element as it's moved around by the velocity field. The first term on the right side of (1) describes the increase (or decrease) of vorticity due to stretching (or compression) of vortex lines. These terms are the mathematical representation of Helmholtz's first and second vortex theorems.

The last term in (1) is the term describing the viscous diffusion of vorticity. For ideal fluids, this term isn't present. The Helmholtz theorems, and Kelvin's circulation theorem, were formulated for ideal fluids, so the viscous term somehow makes these theorems invalid. We discussed why this is in the context of Helmholtz's second theorem. That theorem led to the conclusion that stretching vortex tubes increases their vorticity proportionally, while compressing them decreases the vorticity proportionally. We pointed out that this is analogous to the conservation of angular momentum. Angular momentum is the moment of inertia times the angular velocity, where angular velocity is related to vorticity. By incompressibility, stretching a fluid element makes it thinner, which reduces its moment of inertia. To conserve the angular momentum, the vorticity increases proportionally to the decrease in moment of inertia. For a viscous fluid, however, we can't say that the angular momentum of a vortex tube is conserved, because vorticity can diffuse due to viscosity and thus be transferred out of or into a vortex tube. So Helmholtz's second theorem no longer holds. This viscous diffusion is like a frictional loss. Similar arguments explain the failure of the first Helmholtz theorem and Kelvin's theorem for viscous fluids.

The book talks about the interesting example of the *Burgers vortex*. In Chapter 2 we worked the example of the viscous diffusion of a vortex line. The Burgers vortex is the superposition of a vortex line with a radially inward flow that is, by incompressibility, also extensional in the z -direction. The velocity field is given by:

$$\begin{aligned} u_r &= -\frac{1}{2} \alpha r \\ u_z &= \alpha z \\ u_\theta &= \frac{\Gamma}{2\pi r} \left(1 - e^{-\alpha r^2/4\nu} \right). \end{aligned} \quad (2)$$

The corresponding vorticity field is given by

$$\omega = \frac{\alpha \Gamma}{4\pi \nu} e^{-\alpha r^2/4\nu} \hat{e}_z.$$

1 The Prandtl-Batchelor theorem

Back in Chapter 1, we saw that for 2-D, steady flow of an ideal fluid, the vorticity, ω , was constant along streamlines. We got this result by looking at the vorticity equation for ideal fluids. Another way is to remember that for 2-D, incompressible flow, the velocity field can be written

$$u = \frac{\partial \psi}{\partial y}, \quad v = -\frac{\partial \psi}{\partial x}, \quad (3)$$

where ψ , the stream function, is constant along streamlines. Since the vorticity is $\omega_z = \partial v/\partial x - \partial u/\partial y$, (3) means we can write

$$\omega = \omega_z(\psi),$$

that is, the vorticity is a function of the stream function value, which means the vorticity is constant on a given streamline.

For the ubiquitous case of 2-D flow around a wing, we used this result to conclude that the flow was irrotational everywhere because the flow was irrotational far upstream, and all streamlines started out far upstream. But what if there is a region of closed streamlines in the flow?

If we consider ideal fluid, with $\nu = 0$, then there's nothing we can say about the vorticity in such a region beyond that the vorticity is constant along streamlines. However, if we allow the viscosity to be nonzero but still small, i.e. $\nu \rightarrow 0$, then we can get somewhere. In particular, we have the *Prandtl-Batchelor theorem*, which states that *in steady, 2-D viscous flow the vorticity is constant throughout any region of closed streamlines in the limit $\nu \rightarrow 0$* . The book goes through the derivation of this.

This is a good example of a phenomenon we've discussed occasionally, namely that considering a fluid with no viscosity is not the same as considering a fluid with vanishingly small viscosity.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 17 - The Navier-Stokes equations: velocity gradient tensor

Reading: Acheson, §6.1, 6.2.

Up to now we've discussed the Euler equations that describe the motion of ideal fluids, and the Navier-Stokes equations for viscous, incompressible fluids, without really explaining where they came from. In fact the Navier-Stokes equations in particular took a while to be 'finalized,' and there are some important assumptions inherent in them. We're going to take some time to develop Navier-Stokes equations from the ground up. The discussion will *not* follow the book directly, but the notes and the text are intended to complement each other.

Near the beginning of the course we introduced the velocity gradient tensor, $\nabla \mathbf{u}$ –

$$\nabla \mathbf{u} = \frac{\partial u_j}{\partial x_i} = \begin{pmatrix} \frac{\partial u}{\partial x} & \frac{\partial v}{\partial x} & \frac{\partial w}{\partial x} \\ \frac{\partial u}{\partial y} & \frac{\partial v}{\partial y} & \frac{\partial w}{\partial y} \\ \frac{\partial u}{\partial z} & \frac{\partial v}{\partial z} & \frac{\partial w}{\partial z} \end{pmatrix}. \quad (1)$$

At the time we mentioned this because it represents a vector (the ∇ operator) operating on another vector (the velocity, \mathbf{u}) and yielding a second-order tensor. It turns out the velocity gradient tensor is central to the derivation of the N-S equations. In this lecture we will discuss the physical implications of $\nabla \mathbf{u}$.

1 Deformation, strain, and rotation

As we've seen (and as you recall from your introductory fluids class), in a parallel flow of a Newtonian fluid, the velocity gradient normal to the flow direction is directly proportional to the imposed shear stress. That is, if the velocity is $\mathbf{u} = u(y)\hat{e}_x$, the shear stress $\tau = \mu(du/dy)$, where the proportionality constant μ we've called the dynamic viscosity. Figure 1 illustrates this for the case of a flow between two parallel plates.

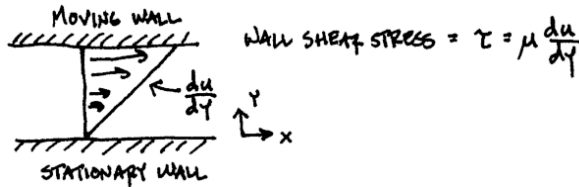


Figure 1: Shear stress in a parallel flow.

Physically (without being precise), we say that the viscosity of the fluid resists deformation because of internal friction. In this example, the 'deformation' takes the form of the shear term du/dy . What about the more general case of non-parallel flow?

For simplicity, we'll talk first about two-dimensional flow, where the velocity field is $\mathbf{u}(x, y) = u(x, y)\hat{e}_x + v(x, y)\hat{e}_y$, and the velocity gradient tensor is

$$\nabla \mathbf{u} = \frac{\partial u_j}{\partial x_i} = \begin{pmatrix} \frac{\partial u}{\partial x} & \frac{\partial v}{\partial x} \\ \frac{\partial u}{\partial y} & \frac{\partial v}{\partial y} \end{pmatrix}. \quad (2)$$

A priori we might expect that the off-diagonal terms of $\nabla \mathbf{u}$, $\partial v/\partial x$ and $\partial u/\partial y$, describe the shear stress, in analogy with the parallel flow case, but it's not quite so simple.

Consider a two-dimensional fluid element, with infinitesimal side lengths $dx = dy$ (Fig. 2a), in

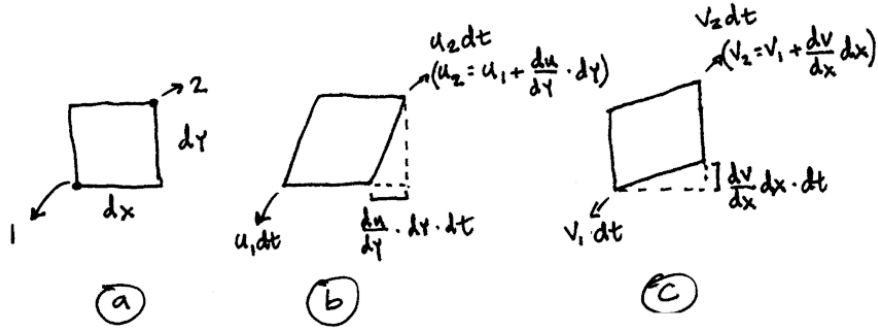


Figure 2: Shear deformation of a 2-D fluid element in parallel flow.

a two-dimensional, steady (not time-varying) flow field. We are not interested now in expansion or compression terms, only shearing terms, so $u(x, y) = u(y)$ and $v(x, y) = v(x)$. (This is equivalent to looking only at the off-diagonal terms of the velocity gradient tensor.) First, consider the parallel velocity field $\mathbf{u} = u(y)\hat{e}_x$. If the fluid element is square at time $t = 0$, as in Fig. 2a, then some (infinitesimal) time later ($t = dt$) point 1 will have moved a distance $u_1 dt$, and point 2 a distance $(u_1 + (du/dy) dy)dt$, and the fluid element gets deformed as in Fig. 2b. (The side lengths $dx = dy$ are infinitesimal, so we only have to worry about linear velocity variations.)

The fluid element is deformed similarly if the velocity field is parallel in the y -direction, $\mathbf{u} = v(x)\hat{e}_y$. In that case, the fluid element will look like Fig. 2c at time $t = dt$. Since the u and v velocity components are orthogonal, for the 2-D velocity field $\mathbf{u} = u(y)\hat{e}_x + v(x)\hat{e}_y$ we can superpose the two deformations to find the shape of the fluid element at time $t = dt$, as in Fig. 3. In the

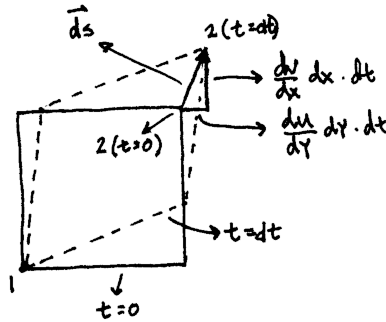


Figure 3: General shear deformation of a 2-D fluid element.

figure, the displacement vector for point 2 is labeled \mathbf{ds} , and can be written

$$\mathbf{ds} = \left(\frac{du}{dy} dy \hat{e}_x + \frac{dv}{dx} dx \hat{e}_y \right) dt = \left(\frac{du}{dy} \hat{e}_x + \frac{dv}{dx} \hat{e}_y \right) dt dx$$

making use of $dx = dy$.

There are two special cases of this shear deformation. If $du/dy = -dv/dx$, then the fluid element rotates, but remains square (Fig. 4). This corresponds to solid body rotation. In this case, there is actually no ‘deformation’ of the fluid element *and no viscous stress*.

The second special case is where $du/dy = dv/dx$ (Fig. 5). In this case the diagonals of the fluid element maintain their angular orientation. This is called pure shear strain, in which there is *no rotation*.

(Acheson has a useful definition of rotation, namely that there is local rotation in a flow if two initially perpendicular fluid line elements have a non-zero average angular velocity. For the cases in Figs. 4 and 5, this means that we just have to see if the diagonals in the fluid element are rotating.)

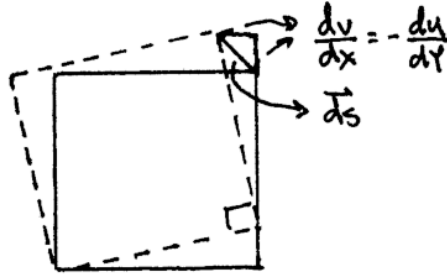


Figure 4: Pure rotation of a 2-D fluid element.

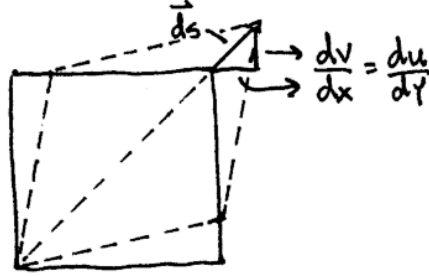


Figure 5: Pure strain of a 2-D fluid element.

Now let's go back and look at only the on-diagonal terms in the 2-D velocity gradient tensor (Eq. 2). Let $\mathbf{u} = u(x)\hat{e}_x + v(y)\hat{e}_y$. The effect of the velocity gradient terms du/dx and dv/dy on the square fluid element of Fig. 2 is obvious; positive du/dx stretches the element in the x -direction, and positive dv/dy stretches the element in the y -direction. Similarly, negative du/dx and dv/dy represent compressions in the x - and y -directions. These are called *normal* strains, and obviously involve no rotation. Viscous fluids resist these normal strains also, but not in quite as intuitive a way as the shear strains (more about this later).

We have now resolved the general 'deformation' of a fluid element under the action of velocity gradients, and have shown that viscosity is relevant to those terms that are independent of rotation. For convenience, then, we want to separate the velocity gradient tensor into parts that describe the strain and rotation separately.

The way to do this suggests itself if we appeal to the above discussion on shear deformations. We saw that there is no viscous stress for $du/dy = -dv/dx$, and no rotation for $du/dy = dv/dx$. The logical thing to do, then, is to make the viscous shear stress proportional to $(du/dy) + (dv/dx)$, and the rotation proportional to $(du/dy) - (dv/dx)$. This is the same thing as separating the tensor $\nabla\mathbf{u}$ into its symmetric and anti-symmetric parts. We'll call the symmetric part \mathbf{e} and the anti-symmetric part Ω –

$$\begin{aligned} \mathbf{e} &= \frac{1}{2} (\nabla\mathbf{u} + (\nabla\mathbf{u})^T) \quad \text{"rate-of-strain tensor"} \\ \Omega &= \frac{1}{2} (\nabla\mathbf{u} - (\nabla\mathbf{u})^T) \quad \text{"rotation rate tensor"} \end{aligned} \quad (3)$$

The factor of 1/2 is incorporated so $\mathbf{e} + \Omega = \nabla\mathbf{u}$. These tensors are called 'rates' of strain and rotation because the terms have units $(\text{time})^{-1}$.

Going back to three dimensions, \mathbf{e} can be written (using symmetry)

$$\mathbf{e} = \begin{pmatrix} e_{11} & e_{12} & e_{13} \\ e_{12} & e_{22} & e_{23} \\ e_{13} & e_{23} & e_{33} \end{pmatrix}, \quad \text{where } e_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (4)$$

and Ω can be written (using anti-symmetry)

$$\Omega = \begin{pmatrix} 0 & \Omega_{12} & \Omega_{13} \\ -\Omega_{12} & 0 & \Omega_{23} \\ -\Omega_{13} & -\Omega_{23} & 0 \end{pmatrix}, \text{ where } \Omega_{ij} = \frac{1}{2} \left(\frac{\partial u_j}{\partial x_i} - \frac{\partial u_i}{\partial x_j} \right) \quad (5)$$

The tensor \mathbf{e} has six independent components, and Ω three (not surprising, since $\nabla \mathbf{u}$ has nine). Since Ω only has three independent components, we might expect that the rotation rate can be expressed as a vector. Recalling the definition of a curl from the first lecture, we write

$$\begin{aligned} \omega = \nabla \times \mathbf{u} &= \begin{vmatrix} \hat{e}_1 & \hat{e}_2 & \hat{e}_3 \\ \frac{\partial}{\partial x_1} & \frac{\partial}{\partial x_2} & \frac{\partial}{\partial x_3} \\ u_1 & u_2 & u_3 \end{vmatrix} \\ &= \left(\frac{\partial u_3}{\partial x_2} - \frac{\partial u_2}{\partial x_3} \right) \hat{e}_1 + \left(\frac{\partial u_1}{\partial x_3} - \frac{\partial u_3}{\partial x_1} \right) \hat{e}_2 + \left(\frac{\partial u_2}{\partial x_1} - \frac{\partial u_1}{\partial x_2} \right) \hat{e}_3 \\ &= \begin{pmatrix} \frac{\partial u_3}{\partial x_2} - \frac{\partial u_2}{\partial x_3} \\ \frac{\partial u_1}{\partial x_3} - \frac{\partial u_3}{\partial x_1} \\ \frac{\partial u_2}{\partial x_1} - \frac{\partial u_1}{\partial x_2} \end{pmatrix} \end{aligned} \quad (6)$$

Comparing (6) with (5), we see that $2\Omega_{ij} = \omega_k \epsilon_{ijk}$, or

$$\Omega = \frac{1}{2} \begin{pmatrix} 0 & \omega_3 & -\omega_2 \\ -\omega_3 & 0 & \omega_1 \\ \omega_2 & -\omega_1 & 0 \end{pmatrix}. \quad (7)$$

The quantity $\omega = \omega_1 \hat{e}_1 + \omega_2 \hat{e}_2 + \omega_3 \hat{e}_3$ is just the vorticity. We already discussed a little how the vorticity is analogous physically to the angular velocity, or rotation rate. Equation (7) tells us the same thing mathematically.

2 Normal strains and viscosity

Let's look back at the on-diagonal terms of (4):

$$\begin{aligned} e_{11} &= \frac{\partial u_1}{\partial x_1} \\ e_{22} &= \frac{\partial u_2}{\partial x_2} \\ e_{33} &= \frac{\partial u_3}{\partial x_3}. \end{aligned}$$

In Cartesian coordinates, these are just $\partial u/\partial x$, $\partial v/\partial y$, and $\partial w/\partial z$, describing expansion and compression. Notice that, for incompressible flows,

$$e_{ii} = e_{11} + e_{22} + e_{33} = \nabla \cdot \mathbf{u} = 0.$$

The sum of the diagonal terms of a tensor is known as its *trace*. For incompressible flows, then, the trace of the rate-of-strain tensor is zero. (This will become interesting later.)

To summarize: the physical reason for separating $\nabla \mathbf{u}$ into the rate-of-strain and rotation rate tensors in (3) is because of the effects of viscosity. The components of the rate-of-strain tensor \mathbf{e} describe motions that are resisted by viscosity; shearing, compression and expansion. The components of the rotation rate tensor Ω describe motions that are not resisted by viscosity. We will incorporate this knowledge into the N-S equations in the next lecture.

3 Further reading

Discussions similar to the above can be found in

- White (on reserve), Section 1-3.3. This treatment is not particularly logical mathematically.
- Batchelor (on reserve), Section 2.3. This treatment is very (maybe too) mathematical.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 18 - The Navier-Stokes equations: stress tensor

Reading: Acheson, §6.1-4.

In the last class we learned that the deformation of a fluid element is described by the strain rate tensor, \mathbf{e} , which is the symmetric part of the velocity gradient tensor, $\nabla \mathbf{u}$ –

$$\mathbf{e} = \frac{1}{2}(\nabla \mathbf{u} + \nabla \mathbf{u}^T).$$

We can then write \mathbf{e} as

$$\mathbf{e} = \begin{pmatrix} e_{11} & e_{12} & e_{13} \\ e_{21} & e_{22} & e_{23} \\ e_{31} & e_{32} & e_{33} \end{pmatrix}, \text{ where } e_{ij} = e_{ji} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right). \quad (1)$$

The significance of this is that, in a viscous fluid, the straining or deforming motions described by \mathbf{e} are precisely the motions that are resisted by viscosity. In order to derive the Navier-Stokes equations, we need to describe the forces that act on a fluid element, and to do that, we will need to incorporate this strain rate tensor.

1 The stress tensor

The ideas we apply to derive the Navier-Stokes equations are pretty general and, in particular, similar ideas pop up in solid mechanics, especially the ideas behind the relationship between stresses, by which we mean forces acting over surfaces, and strains, by which we mean geometric deformations.

What we want to do is describe the forces acting on the surfaces of a volume element of fluid. As usual, we will consider a cubic volume element in Cartesian coordinates, and also as usual, the side lengths will be infinitesimal, so any flow quantities (like velocity) will be essentially constant over each face. Obviously, there will be a pressure force on each face i equal to $p_i dS_i$, where p_i

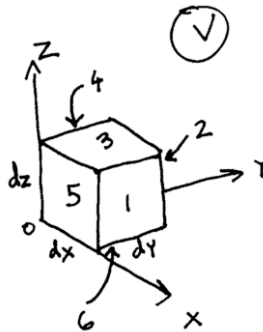


Figure 1: Our usual cubic volume element.

is the pressure on the face i and dS_i is the area of the face. Since we are interested in the forces acting *on* the volume element by the fluid outside of the element, the pressure force points normal to each face and towards the inside of the element.

There will also be viscous forces acting on each face i . Suppose we are talking about face 1 in Fig. 1. On face 1, there are three possible viscous stress terms, as depicted in Fig. 2. There is a normal viscous stress if $du/dx \neq 0$. If du/dx is positive, the force acting on the fluid element across face 1 is in the positive x -direction, as shown. There are also two shear viscous stresses,

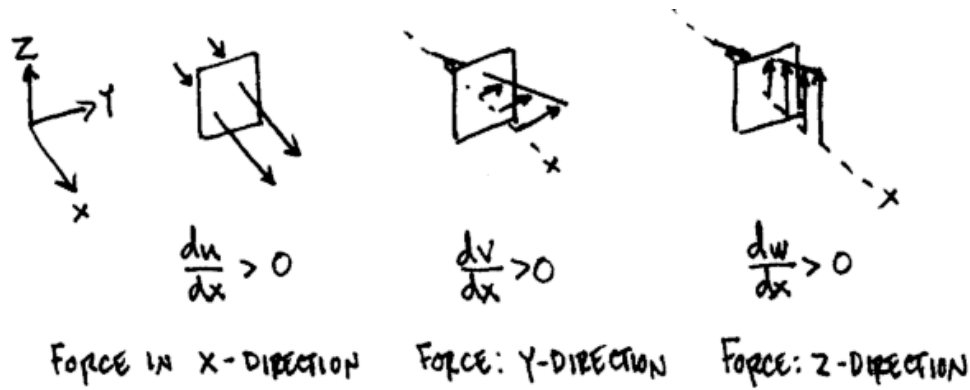


Figure 2: Viscous forces acting on face 1 of the volume element in Fig. 1.

shown in the figure as simple parallel flow shearing motions. If dv/dx is positive, there is a shear force acting on the fluid element across face 1 in the positive y -direction. If dw/dx is positive, the resulting shear force across face 1 is in the positive z -direction. (If these velocity gradient terms are negative, these forces are clearly in the appropriate negative directions.)

Given this multitude of force terms on each face, the handy way to keep track of them is as follows. We define the *stress tensor*, \mathbf{T} , such that

$$\mathbf{T} = \begin{pmatrix} T_{11} & T_{12} & T_{13} \\ T_{21} & T_{22} & T_{23} \\ T_{31} & T_{32} & T_{33} \end{pmatrix}, \text{ where}$$

- T_{ij} is the i -component of stress acting on a surface whose normal vector is in the j -direction. Thus, if the area of the surface is dS , the i -component of force acting on the surface is $T_{ij}dS$.

We can make this definition a bit more general. For a planar surface with arbitrary orientation, i.e. a planar surface whose normal vector $\hat{\mathbf{n}}$ is not aligned with one of the coordinate axes, so that $\hat{\mathbf{n}} = n_j \hat{\mathbf{e}}_j$, we can say:

- T_{ij} is the i -component of stress acting on the surface that is associated with the j -component of the normal vector $\hat{\mathbf{n}}$ of that surface. Thus, the net i -component of force acting on the surface is

$$T_{ij}n_j dS \quad (2)$$

(using the implied summation convention).

As is evident from the above definition(s), \mathbf{T} has *two* directions associated with it. This points up one of the really confusing aspects of (second-order) tensors as opposed to vectors – vectors have a single direction associated with them, but second-order tensors have two. In (2), we are basically writing the i -th row of the stress tensor \mathbf{T} , which is the part of \mathbf{T} that describes forces acting in the i -direction, as a vector \mathbf{T}_i , which is thus written

$$\mathbf{T}_i = T_{ij} \hat{\mathbf{e}}_j$$

so that (2) becomes

$$T_{ij}n_j dS = \mathbf{T}_i \cdot \hat{\mathbf{n}} dS.$$

Before we concern ourselves with the exact mathematical form of \mathbf{T} , let's look at how \mathbf{T} fits into the equation of motion for a fluid element.

2 Cauchy's equation of motion

Consider an arbitrary fluid element, enclosed by a surface S . The net force in the i -direction acting on the (entire) surface is

$$\int_S \mathbf{T}_i \cdot \hat{\mathbf{n}} dS = \int_S T_{ij} n_j dS = \int_V \nabla \cdot \mathbf{T}_i dV = \int_V \frac{\partial T_{ij}}{\partial x_j} dV$$

where we have applied the divergence theorem. As we've done before, we can let the arbitrary fluid element be of infinitesimal size, which lets us write

$$\int_V \frac{\partial T_{ij}}{\partial x_j} dV = \frac{\partial T_{ij}}{\partial x_j} dV \quad (3)$$

since the integrand is basically constant over the infinitesimal element. Meanwhile, the gravitational force on the fluid element is

$$\rho g_i dV \quad (4)$$

where g_i is the component of the gravity vector in the i -direction. (We are not constraining the axes so that one of them is necessarily vertical.)

The forces described by (3) and (4) are just equal to the mass of the fluid element times the i -component of its acceleration, where the mass is ρdV and the acceleration is given by the material derivative of u_i . Thus

$$\rho \frac{Du_i}{Dt} = \frac{\partial T_{ij}}{\partial x_j} + \rho g_i, \quad (5)$$

where we've canceled the common dV terms. There is a little bit more precise derivation of (5) in the book (p.206).

3 Incompressible, Newtonian viscous fluids

We said above that the surface of an element of viscous fluid experiences pressure forces, shear viscous forces, and normal viscous forces. Incorporating the pressure in the stress tensor \mathbf{T} is easy. Obviously the pressure is a normal force, so we have

$$T_{ij} = -p \delta_{ij} + (\text{viscous terms}). \quad (6)$$

The minus sign on the pressure ensures that the pressure force imposed on a fluid element always points *into* the fluid element. The Kronecker delta tells ensures that the pressure is a normal force only, i.e. it appears only when $i = j$.

For the viscous forces, we already showed in the last class that the strain rate tensor (1) describes the motions resisted by viscosity. So we anticipate that the strain rate figures in \mathbf{T} . On top of that, for Newtonian fluids, we said that the stresses were *linearly* proportional to strains, with a proportionality constant ν that we called the dynamic viscosity. This allows us to write (6) as

$$T_{ij} = -p \delta_{ij} + \mu 2e_{ij} = -p \delta_{ij} + \mu \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right). \quad (7)$$

There are some interesting properties of the form of this equation, and in fact you can work backwards and arrive at (7) by assuming these properties and then applying some tensor math (for the interested, the Batchelor book on reserve goes over this in detail). The properties, first enumerated by Stokes, are:

- The stress tensor is a linear function of the velocity gradient components (this is essentially equivalent to the Newtonian assumption).
- The tensor is isotropic, meaning it doesn't care what coordinate system you choose to express it.
- If the fluid isn't deforming, that is if the strain rates are zero, \mathbf{T} reduces to the pressure terms only.

In deriving (7) we have also implicitly assumed that the flows described are incompressible.

The isotropy property is interesting because, for starters, it tells us that the fluid resists stretching and compression in the same quantitative way that it resists shearing. In the equation this is expressed by there being only a single viscosity, ν . Thus the contribution of a normal strain, say $\partial u_1/\partial x_1$, to the stress is $\mu(\partial u_1/\partial x_1)$, while the contribution of a shear strain, say $\partial u_2/\partial x_1$, to the stress is $\mu(\partial u_2/\partial x_1)$. It seems kind of strange that the same coefficient should be valid for both normal and shear stresses.

But then, it seems less strange if we look at a simple example. Consider a two-dimensional velocity field defined as (it's as usual easier to conceive of these motions in two dimensions):

$$u_1 = \beta x_2, \quad u_2 = \beta x_1$$

which corresponds to a pure deformation like that on the left in Fig. 3. Notice that because of the

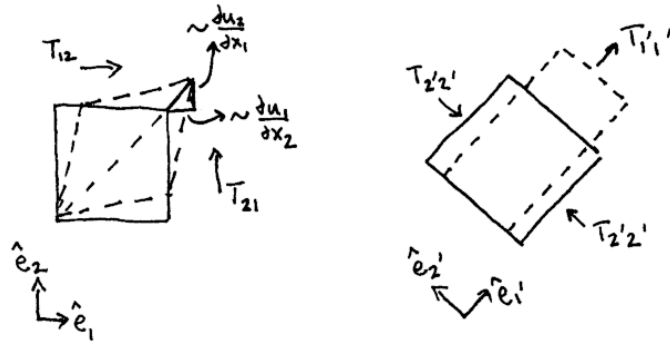


Figure 3: Example of rotation of coordinate systems.

symmetry of the strain rate tensor, the stress components T_{12} and T_{21} are equal (this is another key property of the stress tensor: think about what it implies for a simple parallel shear flow like $\mathbf{u} = (u(y), 0, 0)$). Essentially, what we get is a stretching of the square fluid along the one diagonal, and a compression on the other. Now, what happens if our coordinate system happens to be aligned with those diagonals? Then, we get the situation depicted on the right in Fig. 3. The original shear terms T_{12} and T_{21} correspond to normal terms $T_{1'1'}$ and $T_{2'2'}$ in the new primed axes \hat{e}'_1 and \hat{e}'_2 . So it's clear why only one viscosity coefficient is used to describe shear and normal stresses; because by changes in coordinate systems, you can literally change shear terms into normal terms.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 19 - The Navier-Stokes equations: more on the stress tensor, and kinetic energy transport

Reading: Acheson, §6.4.

At the end of the last lecture we saw why a single proportionality constant, the dynamic viscosity, μ , relates shear strain to shear stresses and also normal strain to normal stresses; the reason is that, just by rotating our coordinate systems, we can actually change shearing motions to compressional or extensional motions. We showed this schematically in terms of a stretching of a two-dimensional, square fluid element along one of its diagonals, which turns into a combination of extension in one axis and compression in the other when we rotate our coordinate system by 45° .

1 Rotation of coordinate systems

What exactly do we mean by rotating our coordinate systems? Consider Fig. 1, depicting a coordinate system with axes \hat{e}_1 and \hat{e}_2 and a second system, obtained by a counter-clockwise rotation through θ , with axes \hat{e}'_1 and \hat{e}'_2 . As can be seen (with some thought) from the figure, for

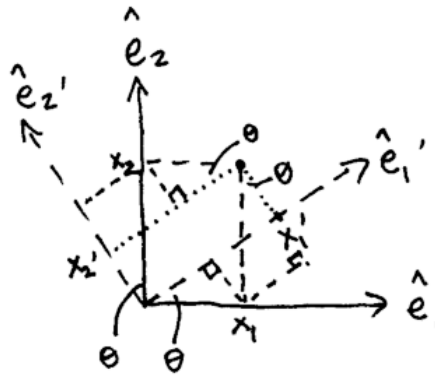


Figure 1: Relation between rotated coordinate systems.

a given point in the plane, the coordinates (x_1, x_2) relate to the coordinates in the rotated system, $(x_{1'}, x_{2'})$, by

$$\begin{aligned}x_{1'} &= x_1 \cos \theta + x_2 \sin \theta \\x_{2'} &= -x_1 \sin \theta + x_2 \cos \theta\end{aligned}$$

which can be expressed in terms of vectors and tensors as

$$\mathbf{x}' = \mathbf{R}\mathbf{x},$$

where

$$\mathbf{x}' = \begin{pmatrix} x_{1'} \\ x_{2'} \end{pmatrix}, \quad \mathbf{R} = \begin{pmatrix} \cos \theta & \sin \theta \\ -\sin \theta & \cos \theta \end{pmatrix}, \quad \mathbf{x} = \begin{pmatrix} x_1 \\ x_2 \end{pmatrix}$$

We can also take the inverse of this rotation, defined as

$$\mathbf{x} = \mathbf{R}^{-1}\mathbf{x}', \tag{1}$$

where

$$\mathbf{R}^{-1}\mathbf{R} = \mathbf{R}\mathbf{R}^{-1} = \mathbf{I}, \quad (2)$$

with \mathbf{I} being the identity matrix, with ones on the diagonal and zeroes everywhere else.

How can we express tensors in these rotated coordinate systems? An easy way to see this is as follows. Often, a tensor will be multiplied by a vector to yield another vector, which can be written:

$$\mathbf{T}\mathbf{x} = \mathbf{y} \quad (3)$$

where \mathbf{T} is a tensor and \mathbf{x} and \mathbf{y} are vectors. The same relationship can be written in the rotated coordinate system $\mathbf{x}' = \mathbf{R}\mathbf{x}$:

$$\mathbf{T}'\mathbf{x}' = \mathbf{y}' \quad (4)$$

where \mathbf{T}' is the original tensor \mathbf{T} expressed in the rotated coordinate system. To relate \mathbf{T} to \mathbf{T}' , we can apply (1) to (3), giving

$$\mathbf{T}\mathbf{R}^{-1}\mathbf{x}' = \mathbf{R}^{-1}\mathbf{y}'.$$

Multiplying both sides by \mathbf{R} , and using (2), we get

$$\mathbf{R}\mathbf{T}\mathbf{R}^{-1}\mathbf{x}' = \mathbf{y}'. \quad (5)$$

And, comparing (5) with (4), we see that

$$\mathbf{R}\mathbf{T}\mathbf{R}^{-1} = \mathbf{T}'.$$

2 Topology of the strain rate field

There is a useful result from linear algebra and tensor math that says that, for any symmetric tensor \mathbf{T} , there is a coordinate rotation represented by \mathbf{R} such that the tensor expressed in the rotated coordinate system, \mathbf{T}' , has non-zero terms only on the diagonal.

This has direct implications for the strain rate tensor, \mathbf{e} . Since \mathbf{e} is symmetric, it can be rotated into a coordinate system where it is diagonal, i.e.

$$\mathbf{e}' = \begin{pmatrix} \alpha & 0 & 0 \\ 0 & \beta & 0 \\ 0 & 0 & \gamma \end{pmatrix}. \quad (6)$$

Since the diagonal terms of the strain rate tensor describe normal strains (extensions and compressions) and the off-diagonal terms describe shears, (6) describes a strain field that has only normal strains and doesn't have shear strains. This formalizes what we discussed in the example at the end of the last lecture, that shear strains can become normal strains by rotating the coordinate system. The diagonal terms in the diagonalized strain rate tensor are called the *principal strain rates*.

We can say even a little more about the strain field. We have also pointed out that the incompressibility condition, $\nabla \cdot \mathbf{u} = 0$, means that the trace of the strain rate tensor (meaning the sum of the terms on the diagonal) has to be zero. This has to be true whatever Cartesian coordinate system we're looking at, independent of rotation. Let these diagonal terms be α , β and γ as in (6), and let us say that α has the highest value and γ the lowest. Since $\alpha + \beta + \gamma = 0$, this means that

$$\alpha > 0, \quad \gamma < 0,$$

(if both α and γ are zero, things are pretty uninteresting, since then the whole strain rate tensor is zero). The question then becomes, what is the sign of the intermediate principal strain rate, β ? The three cases are:

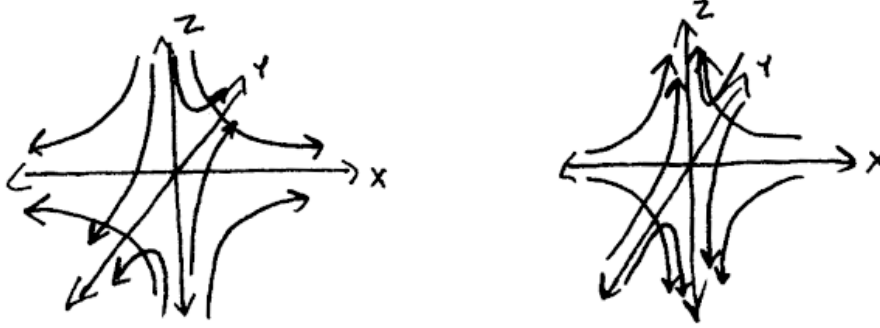


Figure 2: Canonical strain rate topologies: left, positive intermediate principal strain rate, right, negative intermediate principal strain rate.

1. $\beta = 0$: two-dimensional flow, with one extensional principal strain rate and one compressional principal strain rate.
2. $\beta > 0$: this is depicted on the left in Fig. 2. There are two extensional strains and one compressional. This can be viewed as pressing regions of fluid into ‘sheets.’
3. $\beta < 0$: depicted on the right in Fig. 2. There is one extensional strain and two compressional. This can be viewed as stretching regions of fluid into ‘tubes.’

The irrotational parts (that is, the parts described by the strain tensor) of any incompressible velocity field have to fit into one of these three categories. These so-called canonical topologies are widely studied in the context of turbulent flows.

3 Vector form of the Navier-Stokes equation

We have gotten up to the following point with the Navier-Stokes equation:

$$\rho \frac{Du_i}{Dt} = \frac{\partial T_{ij}}{\partial x_j} + \rho g_i, \quad (7)$$

where

$$T_{ij} = -p \delta_{ij} + \mu \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right). \quad (8)$$

Let’s complete the analysis and show how this gives us the form we’ve been using in class up to now.

Combining (7) and (8), we get

$$\begin{aligned} \rho \frac{Du_i}{Dt} &= \frac{\partial}{\partial x_j} \left[-p \delta_{ij} + \mu \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \right] + \rho g_i \\ &= -\frac{\partial p}{\partial x_i} + \mu \frac{\partial}{\partial x_j} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) + \rho g_i \\ &= -\frac{\partial p}{\partial x_i} + \mu \frac{\partial}{\partial x_j} \left(\frac{\partial u_j}{\partial x_i} \right) + \mu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + \rho g_i \end{aligned} \quad (9)$$

Notice how we got rid of the Kronecker delta, δ_{ij} , in the pressure term. Now, looking at the last line, notice that we can interchange the order of partial differentiation in the second term. That is,

$$\frac{\partial}{\partial x_j} \left(\frac{\partial u_j}{\partial x_i} \right) = \frac{\partial}{\partial x_i} \left(\frac{\partial u_j}{\partial x_j} \right).$$

But

$$\frac{\partial u_j}{\partial x_j} = \nabla \cdot \mathbf{u} = 0$$

since the flow is incompressible. Thus (9) becomes

$$\rho \frac{Du_i}{Dt} = -\frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + \rho g_i,$$

which, in vector notation, is just

$$\rho \frac{D\mathbf{u}}{Dt} = -\nabla P + \mu \nabla^2 \mathbf{u} + \rho \mathbf{g}$$

with which we are familiar.

4 Transport of kinetic energy

Acheson talks about the transport of kinetic energy in an interesting fashion (one that I haven't seen in any other texts). The kinetic energy of the fluid in a volume element V enclosed by surface S is:

$$T = \frac{1}{2} \int_V \rho u_i u_i dV$$

(Using T to denote the kinetic energy may seem confusing since we're already using \mathbf{T} for the stress tensor, but the notation is standard in physics. Also, I'm not keen on how Acheson writes u_i^2 instead of $u_i u_i$ – can you think why?) The time rate of change of this T is then

$$\frac{dT}{dt} = \int_V u_i \frac{\partial T_{ij}}{\partial x_j} dV + \int_V \rho u_i g_i dV.$$

In the course of the derivation Acheson has these intermediate steps:

- This step uses the product rule:

$$\int_V u_i \frac{\partial T_{ij}}{\partial x_j} dV = \int_V \frac{\partial}{\partial x_j} (u_i T_{ij}) dV - \int_V T_{ij} \frac{\partial u_i}{\partial x_j} dV$$

- This step uses the divergence theorem:

$$\int_V \frac{\partial}{\partial x_j} (u_i T_{ij}) dV = \int_S u_i T_{ij} n_j dS = \int_S u_i t_i dS$$

where the divergence theorem is applied using the j index as the direction index in the term $u_i T_{ij}$. That is, we treat $u_i T_{ij}$ as the j -component of a vector $\mathbf{u} \cdot \mathbf{T}$. The last equality uses the definition we introduced earlier of the force acting on a surface.

- This step uses the fact of incompressibility:

$$\begin{aligned} \frac{1}{2} T_{ij} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) &= -\frac{1}{2} p \delta_{ij} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \frac{\mu}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)^2 \\ &= -p \frac{\partial u_i}{\partial x_i} + 2\mu e_{ij} e_{ij} = 2\mu e_{ij} e_{ij} \end{aligned}$$

The result is:

$$\frac{dT}{dt} = \overbrace{\int_V \rho \mathbf{u} \cdot \mathbf{g} dV}^{\text{gravitational potential energy}} + \overbrace{\int_S \mathbf{t} \cdot \mathbf{u} dS}^{\text{work done on fluid element}} - \overbrace{2\mu \int_V e_{ij} e_{ij} dV}^{\text{viscous dissipation}}$$

The third term represents viscous losses, which are analogous to friction losses.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 20 - The Navier-Stokes equations: kinetic energy dissipation

Reading: Acheson, §6.5.

At the end of the last lecture we looked at the kinetic energy of the fluid in a volume element, V , enclosed by a surface S ,

$$T = \frac{1}{2} \int_V \rho u_i u_i dV,$$

and we saw that the time evolution of this quantity is given by

$$\frac{dT}{dt} = \overbrace{\int_V \rho \mathbf{u} \cdot \mathbf{g} dV}^{\text{gravitational potential energy}} + \overbrace{\int_S \mathbf{t} \cdot \mathbf{u} dS}^{\text{work done on fluid element}} - \overbrace{2\mu \int_V e_{ij} e_{ij} dV}^{\text{viscous dissipation}}.$$

The first term on the right-hand side represents changes in the gravitational potential energy; the second term represents work done on the surface of the fluid element, by pressure and viscous forces; and the third term represents viscous energy losses. This last term is, obviously, strictly negative, and expresses that whenever there is any deformation of fluid elements in a viscous fluid, energy will be *dissipated* due to viscosity. This viscous kinetic energy dissipation term is neither a conversion of kinetic energy to potential energy nor a work done by the fluid element on the surroundings, but is rather a conversion of kinetic energy to heat. Thus these viscous losses are analogous to friction losses.

There is a very interesting result for the energy dissipation that was first shown by Helmholtz. Before we write that we will take a brief detour into physics.

1 Minimization principles

One of the really attractive ways of thinking about physical problems is in terms of *minimization principles*. An example of these is the refraction of light rays at the interfaces of two media with different indices of refraction. Figure 1 shows a light ray traveling across a plane interface, from a

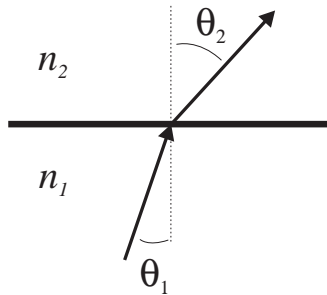


Figure 1: Schematic for Snell's law of refraction.

medium with index of refraction n_1 into one with index n_2 . Define θ_1 as the angle the ray makes in the first medium with the normal to the interface, and θ_2 as the same angle in the second medium. According to Snell's law of refraction, these angles will be related by:

$$\frac{\sin \theta_1}{\sin \theta_2} = \frac{n_2}{n_1}. \quad (1)$$

This is useful enough, but a much more insightful description of refraction (which also works for reflection) is the principle of least time for light rays (due to Fermat, of Last Theorem fame), which states that

- *A light ray always travels from one point to another in a medium by a path that requires the least time.*

We can get from this principle to Snell’s law by remembering that the speed of light in a medium with refractive index n equals c/n , where c is the speed of light in a vacuum. Thus Snell’s law and the principle of least time are equivalent, but clearly they have much different characters. In some sense, Snell’s law is descriptive of the phenomenon of refraction, but the principle of least time is *explanatory*. Think of this another way: Snell’s law is of no use to you if you don’t remember the exact formula (1), but you can always derive the formula if you remember the principle of least time. One consequence of the distinction is that Snell’s law is useless if the refractive index of the medium of interest varies continuously, while the principle of least time can still be used to predict light propagation paths.

There are other well-known minimization principles in physics. Physical bodies seek to minimize their gravitational potential energy, which tells you how to find the shape of a hanging chain. Then there is Hamilton’s principle, which states that the path followed by a dynamical system minimizes the time integral of the difference between the kinetic and potential energies. You can actually use Hamilton’s principle to calculate the equations of motion of a body without using any Newtonian theory, which is pretty remarkable. The question here is: is there any equivalent explanatory principle for fluids?

2 Minimum dissipation theorem

It turns out that there is, but it’s not as general as in the cases of light propagation or dynamical systems. As shown by Helmholtz, for incompressible viscous flow with negligible inertia forces,

- *The actual flow has a smaller total rate of kinetic energy dissipation than any other incompressible flow in the same region having the same values of the velocity vector everywhere on the boundary of the region.*

That is, for given boundary conditions, viscous flows with negligible inertial terms arrange themselves so as to lose the least amount of energy to viscous dissipation. The constraint of negligible inertial terms here means that

$$\frac{\partial \mathbf{u}}{\partial t} \text{ and } (\mathbf{u} \cdot \nabla) \mathbf{u} \ll \text{the viscous and pressure terms}$$

which holds for steady parallel flows, for example (why?). Let’s look at an example.

Consider plane Poiseuille flow of a viscous fluid (Fig. 2). However, we’re going to approach this

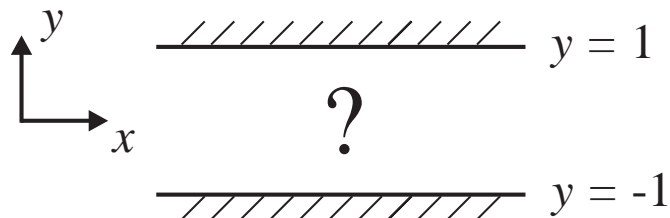


Figure 2: Plane Poiseuille flow.

as if we’ve never heard of the Navier-Stokes equations. Instead we will ask the question—

- Given a fixed mass flow rate per unit span, \dot{m} , between the planes, what is the velocity profile, $u(y)$?

We will make one assumption just for convenience: we will consider only velocity profiles of the form

$$u(y) = A + B|y|^\alpha, \quad (2)$$

where A and B are constants, and α can be any number. What we really want to know is the value of α , which tells us the shape of the velocity profile.

Because the fluid is viscous, the no-slip condition pertains, which automatically tells us

$$A = -B,$$

so we can write $u(y) = A - A|y|^\alpha$. Now let's apply the fixed mass flow assumption, which will give us our value A .

$$\dot{m} = 2\rho \int_0^1 (A - Ay^\alpha) dy,$$

where ρ is the fluid density. Integrating, we get

$$\dot{m} = 2\rho \left[Ay - \frac{1}{\alpha+1} Ay^{\alpha+1} \right]_0^1 = 2\rho A \left[1 - \frac{1}{\alpha+1} \right]$$

so that

$$A = \frac{\dot{m}(\alpha+1)}{2\rho\alpha},$$

and

$$u(y) = \frac{\dot{m}(\alpha+1)}{2\rho\alpha} (1 - |y|^\alpha). \quad (3)$$

Now we want to minimize the kinetic energy dissipation. The dissipation at any point is equal to

$$2\mu e_{ij}e_{ij},$$

but here the only velocity gradient term is du/dy , so the dissipation is $2\mu(du/dy)^2$. We want to minimize this quantity integrated across the region between the planar boundaries. First write the total dissipation:

$$2\mu \int_0^1 \left[-\frac{\dot{m}(\alpha+1)}{2\rho} y^{\alpha-1} \right]^2 dy = \frac{\mu\dot{m}^2(\alpha+1)^2}{2\rho^2} \int_0^1 y^{2\alpha-2} dy.$$

Integrating, we get

$$\frac{\mu\dot{m}^2(\alpha+1)^2}{2\rho^2} \left[\frac{1}{2\alpha-1} y^{2\alpha-1} \right]_0^1 = \frac{\mu\dot{m}^2(\alpha+1)^2}{2\rho^2(2\alpha-1)}.$$

We want the value of α that minimizes this. In particular we'll look for the value of α that minimizes

$$\frac{(\alpha+1)^2}{2\alpha-1}. \quad (4)$$

So we want to set the α -derivative of (4) equal to zero:

$$\frac{d}{d\alpha} \frac{(\alpha + 1)^2}{2\alpha - 1} = \frac{2(\alpha + 1)}{2\alpha - 1} - \frac{2(\alpha + 1)^2}{(2\alpha - 1)^2} = \frac{2\alpha^2 - 2\alpha - 4}{(2\alpha - 1)^2}.$$

The numerator has roots $\alpha = -1$ and $\alpha = 2$. The $\alpha = -1$ solution is unphysical, because the velocity u would go to infinity at $y = 0$. So the proper value of α is 2 (it is trivial to show that this represents a minimum of (4)). This gives us a parabolic velocity profile, which is exactly what we found using the Navier-Stokes equations.

What is the utility of this minimum dissipation formulation? Surprisingly (to me), this concept is never discussed in classes and appears in only one text I'm familiar with (the Batchelor text on reserve). I think it's very useful in that it gives you a clear sense of what the property of viscosity does in a flow. Minimizing dissipation is equivalent to minimizing deformation, which is similar to our prior description that viscosity resists deformations. From a practical standpoint, the minimum dissipation idea is useful in variety of problems including lubrication theory, flow in porous media, etc. (There is one major hurdle, which is that minimization ideas in general need lots of variational calculus....)

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 21 - The energy equation

Reading: Acheson, §8.1-8.2.

In this lecture we will finish up with the equations describing fluid flow. Next, we'll get into boundary layers (note the reading).

1 Energy equation

Acheson doesn't cover it, but the energy equation (more specifically, the internal energy equation) is significant to a lot of engineering problems. To derive it, start by considering the fluid contained in a volume V , enclosed by S . This discussion will repeat some of the ideas we used in deriving the kinetic energy equation, but the distinctions will be clear. Work is done on the fluid element by body forces (gravity) and surface forces. The rate at which work is done on the fluid by gravity is

$$\int_V \rho u_i g_i dV. \quad (1)$$

To interpret this, note that $\rho g_i dV$ is the gravitational force acting on the fluid in an infinitesimal element dV . The work done by that force on a body (in this case, the fluid in dV) is equal to the force times the distance traveled in the direction of the force. Thus the *rate* at which work is done on the fluid in dV , i.e. the work per unit time, is the force times the distance traveled per unit time = the force times the velocity in the direction of the force.

The rate at which work is done on the fluid in V by surface forces is computed similarly. On a given infinitesimal surface element dS , with normal vector $\hat{\mathbf{n}} = n_j \hat{\mathbf{e}}_j$, the surface forces in the i -direction are $T_{ij} n_j dS$ (notice the implied summation over j). The rate at which work is done on the fluid in V by the surface forces on dS is thus

$$u_i T_{ij} n_j dS. \quad (2)$$

Again notice the implied summation over i , which means that (2) represents the sum of the work done by all of the components of the surface forces. Integrating over the entire surface, the rate at which work is done on the fluid in V by surface forces is

$$\int_S u_i T_{ij} n_j dS = \int_V \frac{\partial u_i T_{ij}}{\partial x_j} dV, \quad (3)$$

where we have applied the divergence theorem. To understand this, write the integral on the left side as

$$\int_S (\mathbf{u} \cdot \mathbf{T}) \cdot \hat{\mathbf{n}} dS$$

and note that $\mathbf{u} \cdot \mathbf{T}$ is a *vector* whose components are given by the j index of T_{ij} .

Combining (1) and (3), the rate at which work is done on the fluid in V by all forces is

$$\int_V \left[\rho u_i g_i + u_i \frac{\partial T_{ij}}{\partial x_j} + T_{ij} \frac{\partial u_i}{\partial x_j} \right] dV, \quad (4)$$

where we have expanded (3) using the ubiquitous product rule.

Let's consider just the term in the square brackets in (4), which represents the rate at which work is done on the fluid in the infinitesimal element dV . Recall from a couple of lectures ago that the Navier-Stokes equation can be written

$$\rho \frac{Du_i}{Dt} = \frac{\partial T_{ij}}{\partial x_j} + \rho g_i,$$

so we can write the term in the square brackets in (4) as

$$\text{Rate of working on the fluid in } dV = \rho u_i \frac{Du_i}{Dt} + T_{ij} \frac{\partial u_i}{\partial x_j}. \quad (5)$$

But – the first term on the right can be written

$$\rho u_i \frac{Du_i}{Dt} = \rho \frac{D}{Dt} \frac{1}{2} u_i u_i,$$

which is just the rate of change of kinetic energy (per unit volume). Thus the term

$$T_{ij} \frac{\partial u_i}{\partial x_j} \quad (6)$$

in (5) is the *rate of change of internal energy of the fluid in dV due to work*.

There will also be heat transferred into the fluid by molecular conduction, or diffusion. The rate of heat conduction per unit area is given by $\mathbf{q} = -k\nabla\tau$, where k is the thermal conductivity (which we'll treat as constant) and τ is the temperature. Since \mathbf{q} is a gradient, the rate of heat conduction per unit area in the direction of a vector \mathbf{s} is $\mathbf{q} \cdot \mathbf{s}$. Using this and integrating, the rate of gain of heat by the fluid in V is

$$\int_S k \frac{\partial \tau}{\partial x_i} n_i dS = \int_V k \frac{\partial^2 \tau}{\partial x_i \partial x_i} dV. \quad (7)$$

Two observations. First, the integrand is positive although the \mathbf{q} is defined with a minus sign, because the normal vector $\hat{\mathbf{n}}$ is defined positive *out* of V , and we want to know the heat flux *into* V . Second, we've applied the divergence theorem as usual. From (7), we can write that the *rate of transfer of heat to the fluid in dV due to conduction* is

$$k \frac{\partial^2 \tau}{\partial x_i \partial x_i}. \quad (8)$$

A brief appeal to thermodynamics now. We can write the change in internal energy (per unit mass), E , of a fluid as

$$\Delta E = Q + W,$$

where Q is heat transferred to the fluid and W is the work performed. Here, our Q is given by (8) and our W is given by (6). Thus, we can write the rate of change of internal energy for an infinitesimal fluid element as

$$\rho \frac{DE}{Dt} = T_{ij} \frac{\partial u_i}{\partial x_j} + k \frac{\partial^2 \tau}{\partial x_i \partial x_i}. \quad (9)$$

We can simplify this a bit. The first term on the right can be written

$$\frac{1}{2} T_{ij} \frac{\partial u_i}{\partial x_j} + \frac{1}{2} T_{ji} \frac{\partial u_j}{\partial x_i}$$

since i and j are just dummy variables (we can treat them as such because we sum over them). But \mathbf{T} is symmetric, so this becomes

$$T_{ij} \left(\frac{1}{2} \frac{\partial u_i}{\partial x_j} + \frac{1}{2} \frac{\partial u_j}{\partial x_i} \right) = T_{ij} e_{ij}.$$

We can write this out too:

$$T_{ij} e_{ij} = (-p \delta_{ij} + 2\mu e_{ij}) e_{ij} = -p e_{ii} + 2\mu e_{ij} e_{ij} = 2\mu e_{ij} e_{ij},$$

since $e_{ii} = \nabla \cdot \mathbf{u} = 0$ for incompressible flows. But this is just the kinetic energy dissipation term! Thus, the energy equation is

$$\rho \frac{DE}{Dt} = 2\mu e_{ij} e_{ij} + k \frac{\partial^2 \tau}{\partial x_i \partial x_i}, \quad (10)$$

which says that the internal energy of the fluid increases because of the heat generated by viscous dissipation, and because of thermal conduction. Observe that we have *not* considered radiative heat transfer, or heat release by chemical reactions etc., which are among many effects that can modify this equation.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 22 - Boundary layers: physical insight

Reading: Acheson, §8.1-8.3.

The study of boundary layers occupies a central place in fluid dynamics for a number of reasons. For one, boundary layer theory sits at the intersection of inviscid and viscous flow theory, which is one of the major conceptual divisions in fluids. For another, boundary layers are seen in some form in almost every flow of practical engineering interest. In fact, this is true even in the absence of solid surfaces; the relevant ‘boundary’ can be between, say, high- and low-speed fluid streams.

In general, we will define a boundary layer as follows:

- **Boundary layer:** a thin layer in which the effect of viscosity is important.

Under what conditions will viscosity be important? We know from our discussion of the Navier-Stokes equations that viscosity comes into play when we deform fluid elements, so we can expect that viscosity will be important where velocity gradients are high. When will that happen? As an example, any time we introduce a stationary solid surface into a flow of viscous fluid, a boundary layer will form, because of the no-slip condition. Near the solid surface, the flow needs to adjust itself from the main flow velocity to the velocity of the solid surface. This gives rise to a velocity gradient, which makes viscous effects important.

Just because a boundary layer exists in a flow doesn’t mean that the boundary layer is significant to the flow properties. Think back to the airfoil example we talked about near the beginning of class. We mentioned that at low angles of attack, the lift of a wing is well predicted by ideal flow theory, which ignores viscosity and thus ignores any possible boundary layers. However, at higher angles of attack, ideal flow theory is less able to predict lift. What causes the difference? The answer was *boundary layer separation*, also known as *flow separation*. When the boundary layer is attached to the surface of the wing, it remains thin, and the percentage of the flow volume in which viscous effects are significant is small. When the boundary layer separates, though, the percentage of the flow volume in which viscous effects are significant rises dramatically, and ideal flow theory is inadequate to describe the flow. It’s worthwhile look at boundary layer separation in more detail.

1 Boundary layer separation

A boundary layer can remain attached to a surface if the flow streamlines in the boundary layer have no difficulty following the surface contours. To understand what might cause the boundary layer to be unable to follow the surface contours, consider Fig. 1, depicting the boundary layer on a flat plate, where the mean flow is from left to right. In the leftmost cartoon, the flow is moving in the positive x -direction, pushed along by the negative pressure gradient, $dp/dx < 0$. The velocity gradient at the surface, $du/dy (y = 0)$, is positive. Now suppose we bump up the pressure gradient at the surface. As the pressure gradient changes sign and becomes positive, it will want to push the flow in the negative x -direction. If the flow has some inertia, it will tend to push against this *adverse pressure gradient*. At some point, these effects will be in balance, and the situation in the middle cartoon in Fig. 1 will pertain, where du/dy at the surface is zero. As the pressure gradient is increased past this point, we get flow reversal near the surface (the cartoon on the right). When this happens, the left-to-right boundary layer flow can no longer follow the contour of the surface. Instead, the boundary layer is diverted upward to go around the region of reversed flow. This process is called *boundary layer separation*.

The favorite example of fluids instructors and textbooks everywhere for illustrating boundary layer separation is flow around a sphere. Figure 2 shows schematically what happens if the fluid is

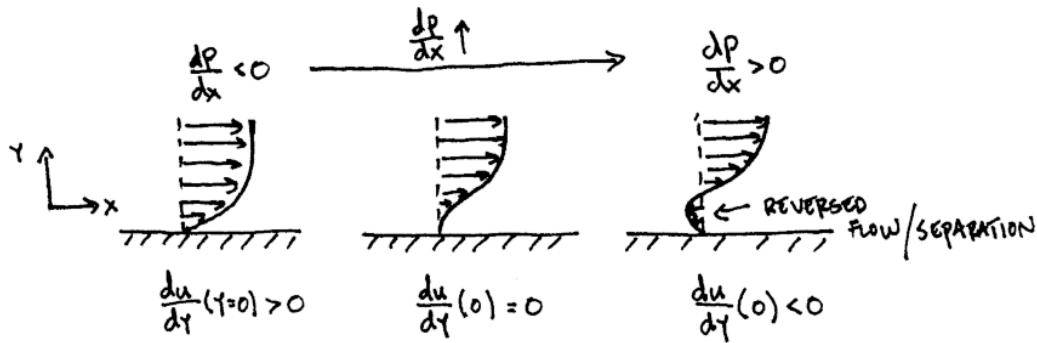


Figure 1: Representation of boundary layer separation.

inviscid. As the flow moves around the front half of the sphere, it accelerates (basically, the fluid has to fit through a smaller space), and by the Bernoulli theorems, the pressure decreases. Then, as the flow moves past the back half of the sphere, the velocity decreases and the pressure increases until both return to their initial values. The velocity and pressure profiles are symmetric between

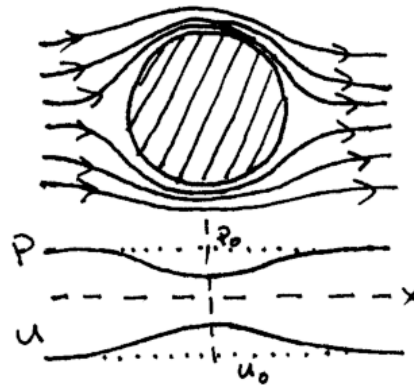


Figure 2: Ideal fluid flow around a sphere.

the front and back of the sphere, because (in the absence of viscosity) there is no friction-type effect to sap energy from the flow. In particular, there is no pressure drag on the sphere (part of what D'Alembert noted in his paradox).

Now let's consider what happens if the viscosity of the fluid is non-zero. This situation is depicted in Fig. 3. There is now a boundary layer at the sphere's surface. On the front half of the sphere, the pressure at the surface is decreasing as we move in the flow direction, so according to the examples of Fig. 1, the boundary layer flows happily along the surface of the sphere. However, as we move to the back half of the sphere, the pressure at the surface *increases* as we move in the flow direction. Eventually, the boundary layer will be unable to fight against this adverse pressure gradient, and the flow will separate. After that, the pressure and velocity profiles essentially freeze; that is, the profiles don't go on to recover to their initial values. The resulting pressure distribution looks something like that shown in Fig. 3. The net pressure on the front of the sphere is higher than that on the back of the sphere, meaning that there is a pressure drag on the sphere (as everyone is amply familiar with in the real world). A good illustration of this boundary layer separation phenomenon is given in Fig. 4.

As is well-known, the pressure drag on a sphere is lowered when the boundary layer flow is *turbulent*. This is because a turbulent boundary layer is better able to overcome the adverse pressure gradient. We will discuss the reasons for this when we discuss turbulent flows later in the semester. The boundary layer can be caused to become turbulent if the sphere's surface is

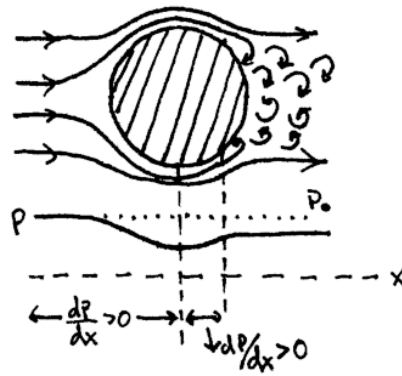


Figure 3: Viscous fluid flow around a sphere.

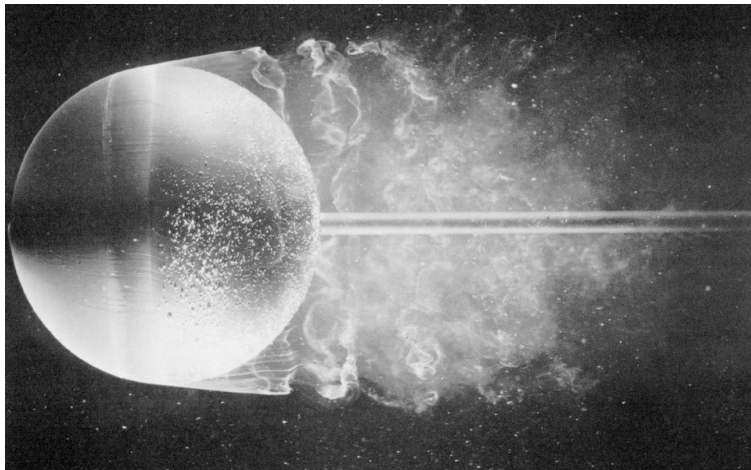


Figure 4: Boundary layer separation on a sphere (from van Dyke, *An Album of Fluid Motion*.)

somehow rough – examples are the dimples on a golf ball or the raised seams on a baseball. In Fig. 5, the boundary layer is made turbulent by a wire placed around the sphere upstream of its equator. By comparing Fig. 5 with Fig. 4, it is clear that the turbulent boundary layer gets further around the sphere before separating. The result is that the pressure recovers better around the back of the sphere, so the pressure on the back of the sphere is higher for the turbulent boundary layer as compared with the laminar boundary layer. Thus, by ‘tripping’ the boundary layer into turbulence, we can reduce the pressure difference between the front and back of the sphere, and we reduce the pressure drag on the sphere.

We might reasonably expect that the same physical reasoning pertains when we talk about lift of bodies (wings, for example) in viscous flows instead of drag. After all, drag arises from the difference in pressure between the front and back of a body, and lift arises from the pressure difference between the top and bottom (Fig. 6). Let’s see if our reasoning regarding drag on a sphere carries over to the lift generated by a wing.

Consider the left cartoon in Fig. 7. In this case, the wing makes an angle of attack with the incoming flow, α , of zero. The boundary layer stays attached over the whole wing, by design. The long, tapered rear to a wing lessens the adverse pressure gradient experienced by the boundary layer, by lengthening the distance over which the pressure recovers. The pressure profiles on the top surface and bottom surface of the wing are also shown in the figure. (Wings are designed so the pressure drop on the top of the wing is much higher than the pressure increase on the bottom surface.)

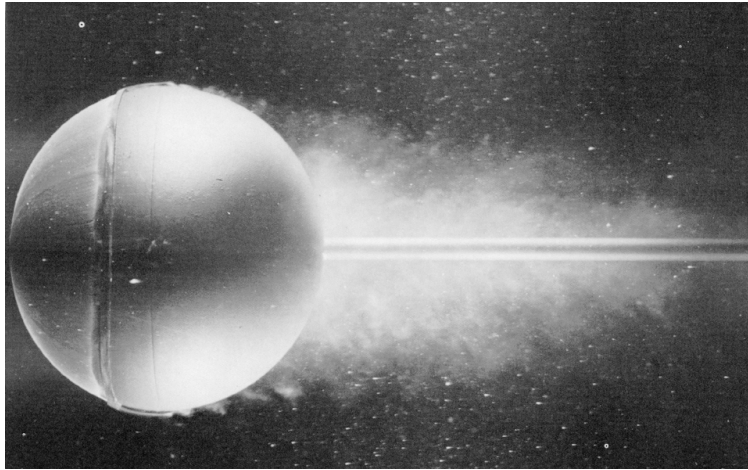


Figure 5: A turbulent boundary layer on a sphere (from van Dyke).

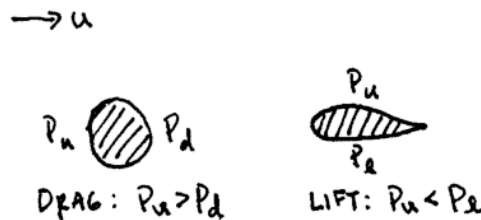


Figure 6: Drag vs. lift.

Now suppose we increase α a little. As we learned in the first chapter of Acheson, as α increases, the lift of the wing initially increases. How does this happen? The middle cartoon in Fig. 7 gives an idea. At increased α , the flow on the wing's upper surface can't stay attached across the whole surface. (What might cause this?) A good photo of this is shown in Fig. 8. If we follow the reasoning we gave for sphere drag, what happens is that after the separation point on the upper surface, the pressure stops rising to reattain its original value, and so the pressure on top of the wing is lower than it would be if there were no separation. Lower pressure on top of the wing means higher lift, as we can observe experimentally.

But...what happens if we keep bumping up the angle of attack, α (the rightmost cartoon in Fig. 7)? This keeps moving the separation point forward, and what *should* happen if our reasoning is still valid is that the pressure on top should keep going down, and the lift should keep going up. But we know that it doesn't. What's going on?

I have seen one text which explains that the separation region on the upper surface of the wing is a region of 'dead air,' where the pressure is (allegedly) roughly equal to the pressure in the undisturbed air stream infinitely far from the wing. According to this explanation, the drop in lift at high angles of attack is because this region of 'dead air' dominates the upper surface of the wing, so the underpressure effect of the top of the wing is drastically reduced. This sounds reasonable, but what about the case where the wing makes an angle of attack of $\alpha = 90^\circ$? This 'dead air' explanation then predicts that there is no drag!

The point is, neither intuitive explanation for the effect of the separation point above the wing is satisfactory. The idea that separation 'freezes' the pressure profile doesn't work when the angle of attack gets high, and the idea that a region of 'dead air' forms doesn't explain the behavior of a wing at low angles of attack and also contradicts our knowledge of drag behind bodies. Math is useful only to an extent here, too; what we really need are *experiments*.

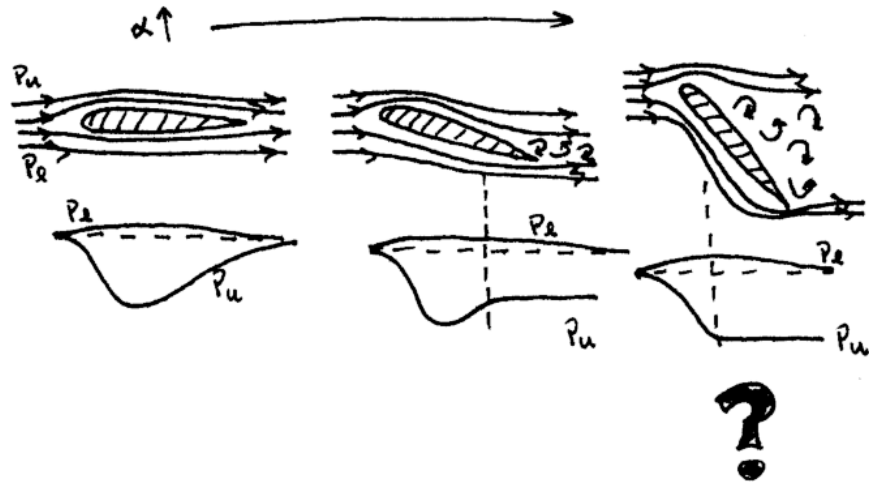


Figure 7: Lift on an airfoil at different angles of attack

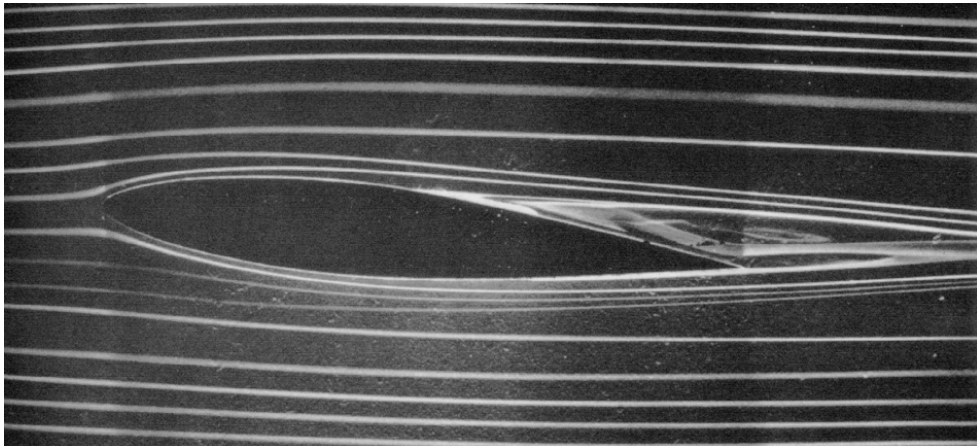


Figure 8: Separation on the upper surface of a wing (from van Dyke).

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 23 - Boundary layers: math basics

Reading: Acheson, §8.2-8.5.

We now want to look at boundary layers in a more rigorous way. It will be helpful to keep in mind exactly what we mean when we speak of boundary layers. In the last lecture we gave the following definition:

- *Boundary layer:* a thin layer in a flow where viscous effects are important.

Implicit in this definition is that viscous effects are *not* important outside of this boundary layer. This is, of course, exactly the situation that we considered with the regard to the problem of the flow of air over a wing. The viscosity of air is low enough that it can effectively be treated as inviscid, *except* near the surface of the wing, where the no-slip condition ensures that viscous forces will be significant.

To help us get a sense of the mathematical approach we'll use, let's look at a sample problem.

1 A sample differential equation

Consider the following problem (starting on p. 269 of the text) concerning a function $u(y)$:

$$\varepsilon \frac{d^2 u}{dy^2} + \frac{du}{dy} = 1 \quad (1)$$

with boundary conditions

$$u(0) = 0, \quad u(1) = 2. \quad (2)$$

The variable ε is a small positive constant.

The general solution to (1) with the given boundary conditions is

$$u(y) = y + \frac{1 - e^{-y/\varepsilon}}{1 - e^{-1/\varepsilon}}. \quad (3)$$

(We'll take this as given without bothering to derive it – it's trivial to verify that (3) satisfies (1) and the boundary conditions.)

Exponentials are very sensitive to the values of their arguments, and in particular the exponential term $e^{-y/\varepsilon}$ is very sensitive to the relative values of y and ε . The solution can be divided into two asymptotic cases. If $y \gg \varepsilon$, then $e^{-y/\varepsilon} \ll 1$, and we get from (3) the main flow (Acheson: 'mainstream') solution

$$u_M(y) = y + 1, \quad (4)$$

while if $y \lesssim \varepsilon$, then $e^{-y/\varepsilon}$ is of order 1, and $y \rightarrow 0$, and we have from (3) the boundary layer solution

$$u_{BL} = 1 - e^{-y/\varepsilon}. \quad (5)$$

Graphically, the solution to (1), and the asymptotic limits (4) and (5), are as shown in Fig. 1 for $\varepsilon = 0.05$. The exact solution (3) matches (5) when $y \lesssim \varepsilon$, which defines the 'boundary layer' in the problem, and (3) matches (4) as y gets large. The boundary layer solution (5) depends on ε , while the main flow solution (4) does not. In a sense, then, the parameter ε is analogous to the viscosity,

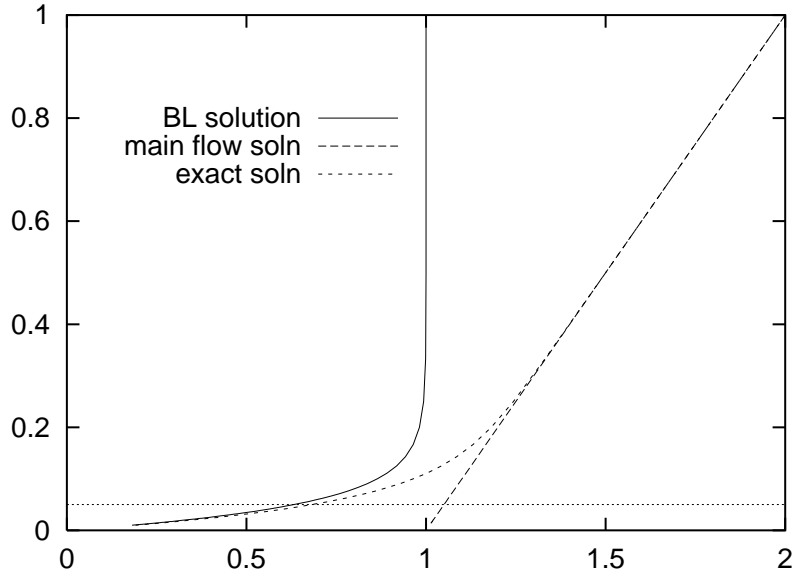


Figure 1: Solution to (1), for $\varepsilon = 0.05$.

because it is significant only in the boundary layer. In fact, the problem defined by (1) is quite a good representation for the Navier-Stokes equations, because this ‘viscosity’ ε is the coefficient of the second derivative in (1), just as the kinematic viscosity ν is the coefficient of the Laplacian in the N-S equations.

In the above discussion, we proceeded by looking at the general solution (3) and identifying different solution regimes based on the relative values of y and ε . We can take an alternate approach, where we determine the asymptotic solutions by looking at the original equation (1). Let’s look for an ‘outer’ solution, $u_o(y)$ by ignoring the ‘viscous’ term $\varepsilon(d^2u/dy^2)$. This is the same thing as letting $\varepsilon \rightarrow 0$ for a given value of y . Our differential equation is then

$$\frac{du_o}{dy} = 1, \tag{6}$$

which is a first-order equation, so we get to use only one boundary condition from (2). The logical one is the boundary condition that $u = 2$ at $y = 1$, since it’s the one in the main or outer part of the flow. The result is that

$$u_o = y + 1. \tag{7}$$

Now we want to find an ‘inner’ solution, u_i , where the viscosity *is* important. The way we proceed is by rescaling our y variable so that the viscous term $\varepsilon(d^2u/dy^2)$ is of the same order as du/dy , the other term involving u in the equation (1), recognizing that ε is small. In other words, we want to rewrite the equation for a new variable

$$Y = y/L, \tag{8}$$

where we need to select the proper value of L . Plugging (8) into (1), we get

$$\varepsilon \frac{1}{L^2} \frac{d^2u_i}{dY^2} + \frac{1}{L} \frac{du_i}{dY} = 1,$$

and obviously the two terms on the left side are of the same order if $L = \varepsilon$, and Y is of order one, which is the same thing as letting $\varepsilon \rightarrow 0$ while $Y = y/\varepsilon$ stays of order one. This is the point of

using the rescaled independent variable Y ; we want to be able to consider values of the independent variable that *aren't* at the limit of extremely small. One benefit to having the two terms on the left side be of the same order is computational – if one term is much bigger than the other, but you don't want to neglect the small one, it's hard to do arithmetic on those terms because your computer only uses a finite number of significant digits.

Anyway, with $Y = y/\varepsilon$, our equation becomes

$$\frac{d^2 u_i}{dY^2} + \frac{du_i}{dY} = 0, \quad (9)$$

which is our boundary layer equation for this problem. The relevant boundary condition from (2) is $u(0) = 0$. By integrating (9), we first get

$$\frac{du_i}{dY} = -u_i + C,$$

and integrating again (separating variables) we get

$$u_i = C + D e^{-Y}.$$

Applying $u_i(0) = 0$, we get

$$u_i = C(1 - e^{-Y}) \quad (10)$$

as our 'inner' solution for the problem. Notice that one free constant, C , still remains.

As our last step, we have to ensure that the outer solution (7) matches with the inner solution (10) at the appropriate place. The appropriate place is for $y \rightarrow 0$ in the outer solution (i.e. as the outer solution approaches the boundary) and for $Y \rightarrow \infty$ in the inner solution (i.e. as the inner solution gets far from the boundary). These limits are

$$u_o \rightarrow 1, \text{ and } u_i \rightarrow C,$$

so $C = 1$. So finally we have

$$u(y) = \begin{cases} y + 1 & \text{for fixed } y \text{ as } \varepsilon \rightarrow 0 \\ 1 - e^{-y/\varepsilon} & \text{for } y/\varepsilon \text{ of order one as } \varepsilon \rightarrow 0, \end{cases} \quad (11)$$

as our solution, which agrees with the equations (4) and (5) we found earlier.

This second method of arriving at the solutions for $u(y)$ in the boundary layer, and in the main flow, is interesting because we didn't need to know the exact form of the solution to equation (1). Instead, we applied boundary layer-type assumptions to simplify (1), and solved the simplified equations. Using this approach to look at fluid flow problems is very useful, because in general we *can't* find exact solutions to the Navier-Stokes equations. What we can do, instead, is assume the flow is inviscid away from boundaries, which gives us the Euler equations, which are easier to solve; near the boundary, we will apply boundary layer assumptions in the hope that we will get simplified equations that we can solve.

2 The 2-D steady boundary layer equations

The exact Navier-Stokes equations for two-dimensional flow are

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right), \quad (12)$$

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right), \quad (13)$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0. \quad (14)$$

We can simplify these considerably for a boundary layer near a wall (say) at $y = 0$. (This is well covered in the text – the discussion here will be sketchy.)

Our basic simplifying assumption is that

- u and v vary much more rapidly in the y -direction than in the x -direction.

One consequence of this can be seen by looking at (14), which is the 2-D continuity equation $\nabla \cdot \mathbf{u} = 0$. We need the terms $\partial u / \partial x$ and $\partial v / \partial y$ to be of the *same order*. Otherwise, u would not vary with x and v would not vary with y (meaning $v = 0$ everywhere), and the problem would be trivialized – our purpose for deriving equations for the boundary layer is to look at their evolution in the x -direction. Since the x -derivative is much smaller than the y -derivative for both u and v , (14) tells us that v is much smaller than u . Now, look at (12) and (13) and compare them term-by-term. We get

$$\left| u \frac{\partial u}{\partial x} \right| \gg \left| u \frac{\partial v}{\partial x} \right|,$$

since $u \gg v$, and

$$\left| v \frac{\partial u}{\partial y} \right| \gg \left| v \frac{\partial v}{\partial y} \right|,$$

again since $u \gg v$. The same applies to the second derivative terms on the right sides. The result is that

$$\left| \frac{\partial p}{\partial x} \right| \gg \left| \frac{\partial p}{\partial y} \right|,$$

(notice: *not* the same behavior as with the velocity components!) which means that

$$p \approx p(x),$$

and we can concern ourselves with the u -equation, (12), and basically ignore the v -equation, (13). Finally, in the u -equation, looking at the right side, we have

$$\left| \frac{\partial^2 u}{\partial y^2} \right| \gg \left| \frac{\partial^2 u}{\partial x^2} \right|,$$

because the y -derivative is much larger than the x -derivative, and this carries over to the second derivatives. So finally we have

$$\begin{aligned} u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} &= -\frac{1}{\rho} \frac{dp}{dx} + \nu \frac{\partial^2 u}{\partial y^2} \\ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0 \end{aligned} \tag{15}$$

as our *2-D boundary layer equations*.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 24 - Boundary layers: the boundary layer on a flat plate

Reading: Acheson, §8.2-8.5.

In Lecture 23 we developed the boundary layer equations for 2-D, steady flow, which were

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{dp}{dx} + \nu \frac{\partial^2 u}{\partial y^2} \quad (1)$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0. \quad (2)$$

To derive these, our key assumption was that u and v vary much more rapidly in the y -direction than in the x -direction. In terms of derivatives, this meant

$$\begin{aligned} \frac{\partial u}{\partial y} &\gg \frac{\partial u}{\partial x} \quad \text{and} \\ \frac{\partial v}{\partial y} &\gg \frac{\partial v}{\partial x}, \end{aligned} \quad (3)$$

which allowed us to toss out various terms in the Navier-Stokes equations. We can get some interesting insight by looking further at (3). Let U_0 be a characteristic value of u , and suppose that u varies by order U_0 over a distance L in the x -direction, and suppose also that u varies by order U_0 over a distance δ in the y -direction. Thus, L is a measure of the boundary layer length and δ is a measure of the boundary layer thickness. In terms of derivatives, we can write

$$\begin{aligned} \frac{\partial u}{\partial y} &= O\left(\frac{U_0}{\delta}\right) \quad \text{and} \\ \frac{\partial u}{\partial x} &= O\left(\frac{U_0}{L}\right), \end{aligned}$$

and by applying (3), we get

$$\delta \ll L, \quad (4)$$

which means that the boundary layer is much thinner than it is long, which we already anticipated earlier in our definition of a boundary layer as a thin layer in where viscous effects are important.

Applying these estimates for the magnitude of u and its derivatives, we can also estimate the magnitude of v . Start with (2):

$$\frac{\partial v}{\partial y} = -\frac{\partial u}{\partial x} = O\left(\frac{U_0}{L}\right). \quad (5)$$

We can approximate $\partial v/\partial y$ in the boundary layer as follows –

$$\frac{\partial v}{\partial y} \approx \frac{v(y = \delta) - v(y = 0)}{\delta}. \quad (6)$$

But $v(y = 0) = 0$ (the boundary is at $y = 0$), so combining (5) and (6), we get

$$v = O\left(\frac{U_0 \delta}{L}\right),$$

which quantifies our qualitative conclusion from the last lecture that v was much smaller than u .

Finally, we can use a similar analysis to estimate the thickness of the boundary layer as a function of the Reynolds number. Looking at (1), we want to ensure that the viscous term on the right side of the equation is comparable in magnitude to the terms on the left. Remember the bigger picture; away from the boundary, the flow is inviscid, so the Euler equation pertains, which means we only have to worry about the left side of (1) and the pressure term on the right. In the boundary layer, viscosity is important, so we additionally have to consider the viscous term on the right side of (1). Applying the same estimates as above, the terms on the left side are each of order U_0^2/L , while the viscous term is of order $\nu U_0/\delta^2$. Setting these magnitudes equal, we get

$$\frac{\delta^2}{L} \approx \frac{\nu}{U_0}, \quad \text{so}$$

$$\frac{\delta}{L} \approx \left(\frac{\nu}{U_0 L} \right)^{1/2} = Re^{-1/2}, \quad (7)$$

where we have defined the Reynolds number as $Re = U_0 L/\nu$. Compare this with (4). Our initial assumption, that u and v vary more quickly in y than in x , led to the condition $\delta \ll L$. From (7), we can see that that assumption is equivalent to requiring that the Reynolds number be large.

1 The boundary layer on a flat plate

Let's look at the example of a boundary layer on a flat plate (Fig. 1). In this situation, a uniform, parallel, incompressible flow with $u = U$ encounters a flat plate, infinitely thin, aligned with the flow direction. The plate is located at $y = 0$ and begins at $x = 0$. The viscosity of the fluid is

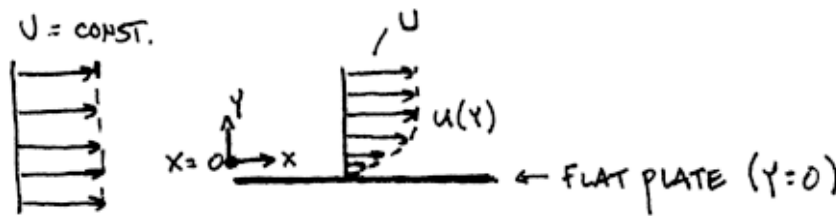


Figure 1: A uniform flow encountering a flat plate.

very small, so the flow is essentially inviscid except near the flat plate, where the no-slip condition pertains, so $u(y = 0) = 0$ for $x > 0$. We will consider the steady-state situation, so time derivatives are zero.

We can make one simplification to the boundary layer equations for this problem. Consider a streamline that is located just 'outside' of the boundary layer. By 'outside,' we mean that the streamline is in that part of the $u(y)$ distribution where $u \approx U$. Because this streamline is outside of the boundary layer, we can treat the fluid as ideal on the streamline. Since the flow is steady, we can apply the (first) Bernoulli theorem on the streamline, i.e. $\frac{p}{\rho} + \frac{1}{2}u^2$ is constant on the streamline. Now, remember from Lecture 23 that we can split the flow into two parts, the main (inviscid) flow and the boundary layer, and get solutions for those parts separately. For this problem, the main flow is uniform, and since the main flow is inviscid, there is no no-slip condition. Thus the main flow remains uniform even in the presence of the plate, and in particular u is constant on streamlines outside the boundary layer. Bernoulli's theorem then tells us that the pressure, p , is constant on those streamlines. Since the streamlines in the main flow are parallel, we have $dp/dx = 0$ in the main flow. But – we saw in the last lecture that p is basically a function only of x in the boundary layer. The pressure distribution inside the boundary layer, then, is the same as the pressure distribution

in the main flow just *outside* the boundary layer. For this problem, this means

$$\frac{dp}{dx} = 0$$

in the boundary layer.

With the pressure term out, then, the boundary layer equations become

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \nu \frac{\partial^2 u}{\partial y^2} \quad (8)$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0. \quad (9)$$

As we've done in other problems, we're going to look for a similarity solution for u . Acheson justifies this approach somewhat in the text; remember (from the review notes for the first exam) that if a similarity solution doesn't exist, it will become obvious in the course of the calculation. The similarity solution we seek is of the form

$$u = Uh(\eta), \quad (10)$$

where

$$\eta = \frac{y}{g(x)},$$

and rather than assume a particular form for the normalizing function g , we'll let it come out of the calculation.

The first thing we'll do is write u (and v) in terms of a stream function, ψ , which will guarantee that our solution automatically satisfies (9). Thus

$$u = \frac{\partial \psi}{\partial y}, \quad v = -\frac{\partial \psi}{\partial x}. \quad (11)$$

By integrating (10) with respect to y , we then get

$$\psi = Ug(x) \int_0^\eta h(s) ds + k(x). \quad (12)$$

Remember now that lines of constant ψ are streamlines. We want the plate to be a streamline (it would be bad if velocity vectors pointed into or out of the plate). We can arbitrarily set $\psi = 0$ on the plate. The surface of the plate is located at $y = 0$, which also means $\eta = 0$. So (12) becomes

$$\psi = Ug(x) \int_0^\eta h(s) ds,$$

which can also be written as

$$\psi = Ug(x)f(\eta), \quad \text{where } f(0) = 0, \quad (13)$$

so $f(\eta)$ is now the similarity function we'll be looking for.

From this form of ψ , and using (11), we have for the velocity components

$$u = Uf'(\eta) \quad \text{and} \quad v = U(\eta f' - f) \frac{dg}{dx} \quad (14)$$

where the primes mean $d/d\eta$. From (14), the velocity derivative terms in (8) are

$$\begin{aligned}\frac{\partial u}{\partial x} &= U f'' \frac{\partial \eta}{\partial x} = U f'' \left(-\frac{y}{g^2} \frac{dg}{dx} \right), \\ \frac{\partial u}{\partial y} &= U f'' \frac{\partial \eta}{\partial y} = U f'' \frac{1}{g} \\ \frac{\partial^2 u}{\partial y^2} &= U f''' \frac{1}{g^2},\end{aligned}\tag{15}$$

and inserting into (8), we get

$$-U^2 f' f'' \frac{y}{g^2} \frac{dg}{dx} + U^2 (\eta f' - f) \frac{dg}{dx} \frac{f''}{g} = \nu U \frac{f'''}{g^2},$$

which gives

$$-U^2 f' f'' \eta \frac{1}{g} \frac{dg}{dx} + U^2 f' f'' \eta \frac{1}{g} \frac{dg}{dx} - U^2 f f'' \frac{1}{g} \frac{dg}{dx} = \nu U \frac{f'''}{g^2},$$

which finally simplifies to

$$f''' + \frac{U}{\nu} g \frac{dg}{dx} f f'' = 0.\tag{16}$$

Remember that in writing (10), we were making the assumption that the velocity, which was originally a function of the two (independent) variables x and y , could instead be expressed as the function of a single independent variable, the similarity variable, η , that combines x and y . Equation (10) led to (13), where the similarity function was $f(\eta)$. In order for the assumption of similarity to be valid, the differential equation for $f(\eta)$, (16), has to have η as the only independent variable. But $g(x)$, which is a function of the independent variable x , appears in (16). So for the similarity assumption to hold, we have to eliminate x as an independent variable. The way to do this is to set $g \cdot dg/dx$ constant. The choice of this constant is arbitrary. We will pick

$$g \frac{dg}{dx} = \frac{\nu}{U}\tag{17}$$

since this choice will also eliminate ν and U from (16).

(Why can we seemingly arbitrarily pick $g(x)$ to satisfy the similarity assumptions? The answer is that we are free to choose g anyway we like – in this case, we’re looking for a similarity solution with the similarity variable $y/g(x)$ – *as long as* the assumptions we make are consistent and our solution for u and v satisfies the boundary layer equations. Basically, the boundary layer equations have a solution. All we’re doing is trying to find that solution. To make the job of finding the solution easier, we express the problem in a more convenient fashion, where in this case ‘convenient’ means in terms of just one independent variable. By doing this we don’t affect the solution in any way. If the solution can’t actually be written in the form we want, we will know because the equations we get will be inconsistent with our assumptions.)

To solve for $g(x)$, first observe that

$$g \frac{dg}{dx} = \frac{d}{dx} \left(\frac{1}{2} g^2 \right).$$

Substituting this in (17) and integrating, we get

$$\frac{1}{2} g^2 = \frac{\nu x}{U} + d$$

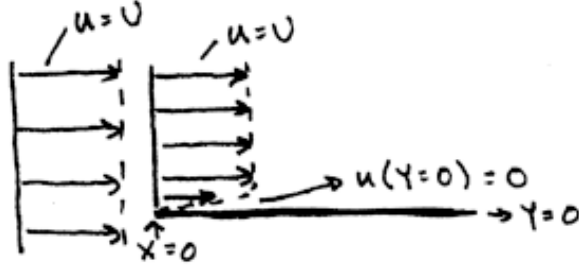


Figure 2: Initial stage of the boundary layer.

where d is a constant. To specify the value of d , look at what happens when $g = 0$. If $g = 0$, then by (15), $\partial u / \partial y$ becomes undefined. But it is quite reasonable for $\partial u / \partial y$ to be undefined at $x = 0$. Consider Fig. 2. Immediately upstream of the plate, at $x \lesssim 0$, the velocity profile $u(y)$ is uniform. Then, starting at $x = 0$, the no-slip condition holds on the plate surface, $y = 0$. For $x \gtrsim 0$, however, the boundary layer is still vanishingly thin, so u has to vary from 0 to U over a distance $\delta \approx 0$. But the derivative $\partial u / \partial y$ is of order U / δ , so at $x = 0$ we can say that $\partial u / \partial y$ is undefined.

We therefore set $g = 0$ at $x = 0$, which gives us

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

or

$$g(x) = \left(\frac{2\nu x}{U} \right)^{1/2}.$$

Finally, we have

$$\psi = (2\nu U x)^{1/2} f(\eta), \text{ where } \eta = \frac{y}{(2\nu x / U)^{1/2}},$$

which satisfies the differential equation

$$f''' + f f'' = 0,$$

with boundary conditions

$$\begin{aligned} f(0) &= f'(0) \\ f'(0) &= 0 \\ f'(\infty) &= 1. \end{aligned}$$

The first boundary condition comes from setting $v = 0$ at $y = 0$ in (14), the second comes from setting $u = 0$ at $y = 0$, and the third is from setting $u = U$ outside the boundary layer.

There is no analytical solution for this problem. Instead, the problem can be solved numerically. The resulting velocity profile, $f(\eta) = u/U$, is shown in Fig. 3.

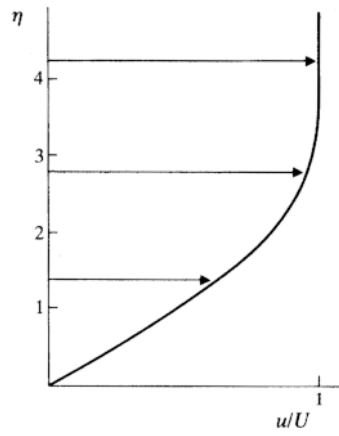


Figure 3: The flat plate boundary layer profile, $u/U = f(\eta)$ (Acheson's Figure 8.8).

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 25 - Boundary layers: high Re flow in a converging channel

Reading: Acheson, §8.3-8.5.

Consider a converging channel, with plane walls located at $y = 0$ and $y = x \tan \alpha$, between which a viscous fluid flows at high Reynolds number. The two walls intersect at the origin, $(x, y) = (0, 0)$. We will assume that there is a slit at the origin through which fluid can either be extracted from the channel (we'll call this the inflow case) or injected into the channel (outflow). First, we want

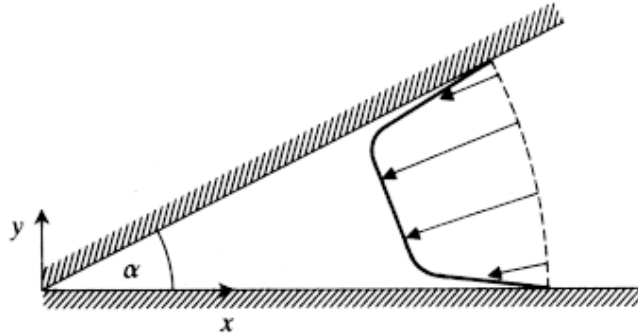


Figure 1: Flow in a converging channel at high Re (Acheson's Fig. 8.9).

to find the inviscid, mainstream flow. Given the geometry of the system, we will look for a radial velocity field, where $\mathbf{u} = u_r(r, \theta)\hat{e}_r$. The continuity condition $\nabla \cdot \mathbf{u} = 0$ becomes:

$$\nabla \cdot \mathbf{u} = \frac{1}{r} \frac{\partial}{\partial r}(ru_r) + \frac{1}{r} \frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z} = \frac{1}{r} \frac{\partial}{\partial r}(ru_r) = 0$$

which gives

$$\frac{\partial}{\partial r}(ru_r) = 0$$

and integrating, we get

$$ru_r = \text{const.}$$

so (calling the integration constant $-Q$ as Acheson does)

$$u_r = -\frac{Q}{r}. \tag{1}$$

Negative Q corresponds to inflow (velocity vectors in the negative r direction), and positive Q corresponds to outflow (velocity vectors in the positive r direction).

With this result, our problem comes down to finding the velocity profile in the boundary layer, $y \gtrsim 0$. (The boundary layer on the other wall will, of course, be symmetric with this one.) With the coordinates defined as in Fig. 1, we can express the flow for $y \approx 0$ in terms of Cartesian coordinates pretty easily. In particular, our velocity field, which we had expressed generally as $\mathbf{u} = u_r(r, \theta)\hat{e}_r$, can be written in the boundary layer as

$$\mathbf{u} = u_x(x, y)\hat{e}_x$$

We now want to establish the proper equations for the boundary layer. The equation for the u -component of velocity is

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{dP}{dx} + \nu \frac{\partial^2 u}{\partial y^2} \quad (2)$$

We saw earlier that the pressure gradient in the boundary layer, dP/dx , can be written in terms of the velocity, $U(x)$, in the inviscid, main flow just *outside* of the boundary layer. In Cartesian coordinates, this main flow is

$$U(x) = -\frac{Q}{x}. \quad (3)$$

(Think carefully about what it means to write (3) in place of (1). The coordinate x is exactly the same as the coordinate r for $y = 0$, and $x \approx r$ for $y \approx 0$. To express the main flow in terms of x , as in (3), the main flow solution must be valid near $y = 0$, meaning the flow must become inviscid for $y \gtrsim 0$. In other words, the boundary layer in this problem is required to be especially thin. As we saw in the last lecture, the ratio δ/L that describes the thickness of the boundary layer is proportional to $Re^{-1/2}$. For this converging channel problem we are therefore assuming that the Reynolds number is high.)

To get the pressure gradient, we use the Bernoulli theorem for steady flow of ideal fluid, which says that $p/\rho + \frac{1}{2}u^2$ is constant along streamlines. Thus, for a streamline just outside the boundary layer,

$$\frac{p}{\rho} = -\frac{1}{2}U(x)^2 + \text{const.} \quad (4)$$

which holds because the flow is parallel in the region of interest. Taking the x -derivative of this equation, we get

$$\frac{1}{\rho} \frac{dp}{dx} = -U \frac{dU}{dx} \quad (5)$$

which, substituting into the boundary layer equation (2), gives

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{Q^2}{x^3} + \nu \frac{\partial^2 u}{\partial y^2} \quad (6)$$

As with the flat plate problem of Lecture 24, we'll look for a similarity solution to this equation, which will be of the form $u = |U(x)|h(\eta)$, where $\eta = y/g(x)$. (By using the absolute value of the main flow velocity, the difference between the inflow and outflow cases is described by the sign of the similarity function.) Using a stream function formulation to satisfy the continuity condition, $\nabla \cdot \mathbf{u} = 0$, we get for the stream function:

$$\psi = |U(x)|g(x)f(\eta) = -\frac{|Q|}{x}g(x)f(\eta) \quad (7)$$

so the velocity components are

$$u = \frac{\partial \psi}{\partial y} = |U|f'(\eta) = -\frac{|Q|}{x}f'(\eta) \quad (8)$$

and

$$v = -\frac{\partial \psi}{\partial x}. \quad (9)$$

(We will hold off on the particular form of v in terms of U and f until later, when it will be simplified considerably.) From (8), we get boundary conditions on f' . Outside the boundary layer, i.e. as $\eta \rightarrow \infty$ (which is *not* the same thing as $y \rightarrow \infty!$), we need $u \rightarrow U(x)$. For the inflow case U is negative, and for the outflow case U is positive, so

$$f'(\infty) = \begin{cases} 1 & \text{in the inflow case} \\ -1 & \text{in the outflow case} \end{cases} \quad (10)$$

Let's look now at (6). If we plug in the expressions for u and v ((8) and (9)) and the appropriate derivatives in terms of the similarity variable, η , then a similarity solution will only be possible if we end up with an ordinary differential equation for $f(\eta)$ (meaning an equation that involves only f , the derivatives of f , and η). But consider the first term on the right side of (6), what was originally the pressure term. It is unchanged by the substitution of the similarity forms. In order for our resulting differential equation to be independent of x , then, we have to multiply each term in (6) by x^3 to eliminate the x -dependence of the pressure term, then we need to require that each of the remaining three terms be independent of x .

The simplest of these three terms to analyze is the second term on the right side of (6) (no messy product rule to deal with). Using (8), we have

$$\begin{aligned} \nu \frac{\partial^2 u}{\partial y^2} &= -\nu \frac{|Q|}{x} \frac{\partial^2 f'(\eta)}{\partial y^2} = -\nu \frac{|Q|}{x} \frac{\partial}{\partial y} \left(\frac{df'}{d\eta} \frac{\partial \eta}{\partial y} \right) = -\nu \frac{|Q|}{x} \frac{\partial}{\partial y} \left(\frac{f''}{g} \right) \\ &= -\nu \frac{|Q|}{x} \frac{f'''}{g^2}. \end{aligned}$$

From the preceding paragraph, we need $x^3 \nu \frac{\partial^2 u}{\partial y^2}$ to be independent of x , which means $g(x)$ has to be proportional to x , for there to be a possible similarity solution. We are free to choose the proportionality constant in the expression for g , and we choose

$$g(x) = \sqrt{(\nu/|Q|)}x$$

because this gives us a 'cleaner' differential equation (fewer constants floating around). In particular, with this choice of g the right side of (6) becomes:

$$-\frac{Q^2}{x^3} + \nu \frac{\partial^2 u}{\partial y^2} = -\frac{Q^2}{x^3} - \frac{|Q|^2}{x^3} f''' = -\frac{Q^2}{x^3} (1 + f''') \quad (11)$$

using $|Q|^2 = Q^2$.

This g also makes computing the terms on the left side of (6) relatively easy, because the stream function ψ in (7) becomes

$$\psi = -\sqrt{\nu|Q|}f(\eta),$$

which gives

$$\begin{aligned} v &= -\frac{\partial \psi}{\partial x} = -\sqrt{\nu|Q|} \frac{\partial f}{\partial x} = -\sqrt{\nu|Q|} f' \frac{\partial \eta}{\partial x} \\ &= -\sqrt{\nu|Q|} f' \left(-\frac{y}{g^2} \frac{dg}{dx} \right) \\ &= -\sqrt{\nu|Q|} f' \left(-\eta \frac{\sqrt{|Q|/\nu}}{x} \sqrt{\nu/|Q|} \right) \\ &= -\frac{\sqrt{\nu|Q|}}{x} \eta f'. \end{aligned}$$

Meanwhile, u is given by (8), and the derivatives of u are

$$\begin{aligned}\frac{\partial u}{\partial y} &= -\frac{|Q|}{x} f'' \frac{\partial \eta}{\partial y} = -\frac{|Q|}{x} \frac{f''}{g} = -\frac{|Q|^{3/2}}{\sqrt{\nu} x^2} f'' \\ \frac{\partial u}{\partial x} &= \frac{|Q|}{x^2} f' - \frac{|Q|}{x} f'' \frac{\partial \eta}{\partial x} = \frac{|Q|}{x^2} f' - \frac{|Q|}{x} f'' \left(-\frac{y}{g^2} \frac{dg}{dx} \right) = \frac{|Q|}{x^2} f' + \frac{|Q|}{x^2} f'' \eta.\end{aligned}\quad (12)$$

Substituting all this into the left side of (6), we have

$$\begin{aligned}u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} &= -\frac{|Q|}{x} f' \left(\frac{|Q|}{x^2} f' + \frac{|Q|}{x^2} f'' \eta \right) - \frac{\sqrt{\nu |Q|}}{x} \eta f' \left(-\frac{|Q|^{3/2}}{\sqrt{\nu} x^2} f'' \right) \\ &= -\frac{Q^2}{x^3} \left(f'^2 + f'' \eta - f'' \eta \right) = -\frac{Q^2}{x^3} f'^2.\end{aligned}\quad (13)$$

Setting (13) and (11) equal, and canceling the common Q^2/x^3 , we get

$$f'^2 = 1 + f''' \quad (14)$$

as our ordinary differential equation for $f(\eta)$. The boundary conditions are given by (10) for $\eta \rightarrow \infty$. For $\eta = 0$ (which means $y = 0$), the boundary condition is the no-slip condition, $u = 0$, so from (8),

$$f'(0) = 0. \quad (15)$$

We can make (14) a second-order, rather than third-order, differential equation by making the substitution $F(\eta) = f'(\eta)$. Then (14) becomes

$$F'' - F^2 + 1 = 0 \quad (16)$$

and the boundary conditions given by (10) and (15) become

$$\begin{aligned}F(0) &= 0, \quad \text{and} \\ F(\infty) &= \begin{cases} 1 & \text{in the inflow case} \\ -1 & \text{in the outflow case.} \end{cases}\end{aligned}\quad (17)$$

1 The inflow case, $F(\infty) = 1$

The analytical solution for this case makes use of a variety of tricks that are non-intuitive. The first thing we do is multiply (16) by F' :

$$F' F'' - F^2 F' + F' = \frac{d}{d\eta} \left(\frac{1}{2} F'^2 \right) - \frac{d}{d\eta} \left(\frac{1}{3} F^3 \right) + \frac{dF}{d\eta} = 0$$

which, integrated, gives

$$\frac{1}{2} F'^2 - \frac{1}{3} F^3 + F = \text{const.} \quad (18)$$

To determine the value of the constant, we just need to know F and F' at one value of η . We know that $F(\infty) = 1$ already. To find $F'(\infty) = f''(\infty)$, look back at $\partial u/\partial y$, given by (12) –

$$\frac{\partial u}{\partial y} = -\frac{|Q|^{3/2}}{\sqrt{\nu} x^2} f'' = -\frac{|Q|^{3/2}}{\sqrt{\nu} x^2} F'. \quad (19)$$

For $\eta \rightarrow \infty$, $u \rightarrow U(x)$, so

$$\frac{\partial u}{\partial y} \rightarrow \frac{\partial U(x)}{\partial y} = 0$$

and by (19),

$$F'(\infty) = 0.$$

Substituting for $F(\infty)$ and $F'(\infty)$ into (18), we get

$$\frac{1}{2}F'^2 - \frac{1}{3}F^3 + F = \frac{2}{3}. \quad (20)$$

Now the sleight of hand starts. We can factor (20) as

$$F'^2 = \frac{2}{3}(1 - F)^2(2 + F). \quad (21)$$

Then the substitution

$$G^2 = 2 + F \quad (22)$$

turns out to be convenient. First, taking the η -derivative of (22), we get

$$2GG' = F'$$

and substituting into (21), we get

$$4G^2G'^2 = \frac{2}{3}(3 - G^2)^2G^2$$

which gives

$$G' = \pm \frac{1}{\sqrt{6}}(3 - G^2) = \pm \frac{3}{\sqrt{6}} \left(1 - \frac{G^2}{3}\right). \quad (23)$$

Our next substitution is $G = \sqrt{3} \tanh H$. We have

$$\begin{aligned} \sinh H &= \frac{e^H - e^{-H}}{2} \quad \text{and} \\ \cosh H &= \frac{e^H + e^{-H}}{2} \end{aligned} \quad (24)$$

giving

$$\begin{aligned} \frac{d}{d\eta}(\sinh H) &= \cosh H \cdot \frac{dH}{d\eta} \\ \frac{d}{d\eta}(\cosh H) &= \sinh H \cdot \frac{dH}{d\eta} \\ \sinh^2 H - \cosh^2 H &= 1 \quad \text{and} \\ \tanh^2 H - 1 &= \operatorname{sech}^2 H \end{aligned}$$

so

$$\frac{d}{d\eta}(\tanh H) = \frac{d}{d\eta} \left(\frac{\sinh H}{\cosh H} \right) = \left(1 - \frac{\sinh^2 H}{\cosh^2 H} \right) \frac{dH}{d\eta} = -\operatorname{sech}^2 H \cdot H'.$$

Thus, with this substitution and (23), we have

$$\begin{aligned}
 G' &= -\sqrt{3} \operatorname{sech}^2 H \cdot H' = \pm \frac{3}{\sqrt{6}} \left(1 - \frac{G^2}{3}\right) \\
 &= \pm \frac{3}{\sqrt{6}} (1 - \tanh^2 H) \\
 &= -\pm \frac{3}{\sqrt{6}} \operatorname{sech}^2 H
 \end{aligned} \tag{25}$$

which gives

$$H' = \pm \frac{1}{\sqrt{2}},$$

and, integrating with respect to η ,

$$H = \pm \frac{\eta}{\sqrt{2}} + C \tag{26}$$

where C is the constant of integration.

Finally, putting all of these substitutions together:

$$F = G^2 - 2 = 3 \tanh^2 H - 2 = 3 \tanh^2 \left(\frac{\eta}{\sqrt{2}} + C \right) - 2, \tag{27}$$

where we've used only the positive solution for H from (26), because $\tanh^2(x) = \tanh^2(-x)$. From (24), $\tanh(\infty) = 1$, so F from (27) satisfies the boundary condition $F(\infty) = 1$. To satisfy the other boundary condition, $F(0) = 0$, we need

$$\tanh C = \pm \sqrt{\frac{2}{3}},$$

and it turns out that the solution to this is

$$C \approx \pm 1.14.$$

Profiles of F vs. η are plotted in Fig. 2, for $C = 1.14$ and $C = -1.14$. Recall from (8) that $u = |U|f' = |U|F$. For positive C , we get the sort of boundary layer depicted in Fig. 1, while for negative C , the boundary layer has a region of reversed flow near the wall. Both of these solutions are obviously valid mathematically; the question of which one would be observed in an actual flow depends on the details of how the flow is set up (more on this in Chapter 9). Also of interest: remember that all the way at the beginning of this analysis we said that the Reynolds number had to be high so that the boundary layer would be thin. Acheson shows in the text how this condition comes down to $Q/\nu \gg 1$ in this case.

2 The outflow case, $F(\infty) = -1$

As mentioned in the text (Exercise 8.6, which is explained in the hints), there is no solution to the boundary layer equations for this case. (Actually, all we can conclude from the book's analysis is that there is no *similarity* solution to the boundary layer equations for this case.)

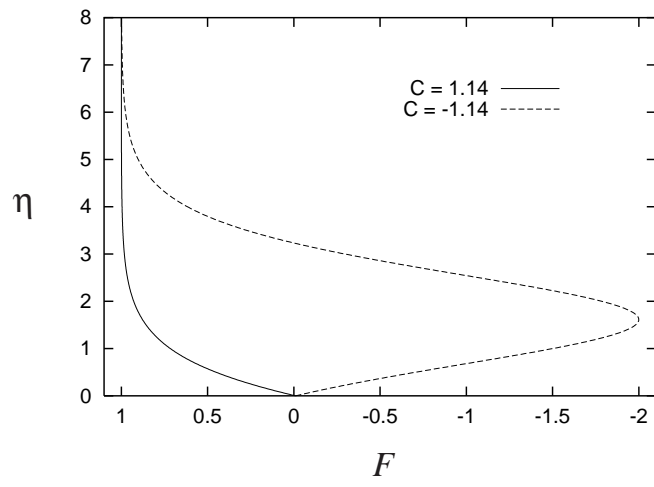


Figure 2: Profiles of $F = u/|U|$ vs. η for the inflow case.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 26 - Boundary layers: equations for rotating flow

Reading: Acheson, §8.5-8.6.

We'll consider now boundary layers in rotating flows. Consider a system with viscous fluid between two rigid boundaries located at $z = 0$ and $z = L$, and suppose the lower boundary rotates around the z -axis with angular velocity Ω , and the upper boundary rotates around the z -axis with angular velocity $\Omega(1 + \epsilon)$, where ϵ is small. The characteristic length scale in this flow is clearly L , and we can define a velocity scale by ΩL , so we can write a Reynolds number for this system as

$$Re = \frac{\Omega L^2}{\nu}.$$

Analogous to our prior examples, if this Reynolds number is large, we expect that viscous effects will be important only in thin layers near the boundaries, and that the flow will be essentially inviscid in the rest of the space. In our previous examples, though, we were able to proceed by determining the inviscid main flow separately, without worrying about the effect of the boundary. For each of those cases, the boundary layer just allowed the given main flow, which was presumed to be driven externally, to match the no-slip boundary condition. In this rotating flow system, the situation is different, because the flow is driven by the moving boundaries themselves.

1 Basic equations

As usual, we'll start by writing the equations for the system. If a fluid is rotating with nearly uniform angular velocity $\vec{\Omega}$, it is helpful to consider a coordinate system that also rotates with angular velocity $\vec{\Omega}$. The Navier-Stokes equations in such a system are

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} + 2\vec{\Omega} \times \mathbf{u} + \vec{\Omega} \times (\vec{\Omega} \times \mathbf{x}) = -\frac{1}{\rho} \nabla P + \nu \nabla^2 \mathbf{u} \quad (1)$$

and

$$\nabla \cdot \mathbf{u} = 0$$

where \mathbf{u} , in this case, is the fluid velocity relative to the moving coordinate system. The third term on the left side of (1) is the Coriolis 'force,' and the fourth term on the left side of (1) is the centrifugal 'force.' These terms only arise because the coordinate system is rotating and thus aren't really true forces. (For example, to understand the centrifugal 'force,' consider what happens to (1) when \mathbf{u} is zero.)

If we apply the vector identity (which we won't bother to prove)

$$\vec{\Omega} \times (\vec{\Omega} \times \mathbf{x}) = -\nabla \left[\frac{1}{2} |\vec{\Omega} \times \mathbf{x}|^2 \right],$$

(note: remember that when Acheson writes \mathbf{F}^2 for some vector \mathbf{F} , he means $\mathbf{F}^2 = \mathbf{F} \cdot \mathbf{F} = |\mathbf{F}|^2$) then we can rewrite (1) in the somewhat simpler form

$$\begin{aligned} \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} + 2\vec{\Omega} \times \mathbf{u} &= -\frac{1}{\rho} \nabla p + \nabla \left[\frac{1}{2} (\vec{\Omega} \times \mathbf{x})^2 \right] + \nu \nabla^2 \mathbf{u} \\ &= -\frac{1}{\rho} \nabla p_R + \nu \nabla^2 \mathbf{u} \end{aligned} \quad (2)$$

where we define the ‘reduced pressure’ as

$$p_R = p - \frac{1}{2}\rho|\vec{\Omega} \times \mathbf{x}|^2.$$

One of our key assumptions here is that \mathbf{u} will be small relative to the overall rotation of the system. For the system we talked about at the start of the lecture, this will be guaranteed because the two boundaries have almost the same angular velocity. If U is a characteristic magnitude of \mathbf{u} , and L is a characteristic length scale of the flow, then we can write

$$|(\mathbf{u} \cdot \nabla)\mathbf{u}| = O\left(\frac{U^2}{L}\right)$$

and

$$|2\vec{\Omega} \times \mathbf{u}| = O(\Omega U)$$

where $\Omega = |\vec{\Omega}|$. The condition that \mathbf{u} is small relative to the overall system rotation means that $U \ll \Omega L$. We can then write

$$\frac{|(\mathbf{u} \cdot \nabla)\mathbf{u}|}{|2\vec{\Omega} \times \mathbf{u}|} = \frac{O(U^2/L)}{O(\Omega U)} = O\left(\frac{U}{\Omega L}\right) \ll 1,$$

or

$$|(\mathbf{u} \cdot \nabla)\mathbf{u}| \ll |2\vec{\Omega} \times \mathbf{u}|,$$

that is, the inertial term in (2) is much smaller than the Coriolis term. Thus, the Navier-Stokes equations for a small velocity \mathbf{u} relative a uniform rotation with angular velocity $\vec{\Omega}$ are

$$\frac{\partial \mathbf{u}}{\partial t} + 2\vec{\Omega} \times \mathbf{u} = -\frac{1}{\rho}\nabla p + \nu\nabla^2 \mathbf{u} \quad (3)$$

$$\nabla \cdot \mathbf{u} = 0. \quad (4)$$

2 Steady, inviscid main flow

Let’s consider now Cartesian coordinates that are fixed in the rotating frame of reference, and let the z -axis be parallel to the rotation axis. Thus

$$\vec{\Omega} = (0, 0, \Omega).$$

We are interested first in the steady, inviscid flow solution for this rotating system. Let this flow be denoted by

$$\mathbf{u}_I = (u_I, v_I, w_I)$$

so that the Coriolis term in (3) is

$$\vec{\Omega} \times \mathbf{u} = \begin{vmatrix} \hat{e}_x & \hat{e}_y & \hat{e}_z \\ 0 & 0 & \Omega \\ u_I & v_I & w_I \end{vmatrix} = -\Omega v_I \hat{e}_x + \Omega u_I \hat{e}_y + (0)\hat{e}_z.$$

Because the flow is steady and inviscid, we can ignore the $\partial \mathbf{u}/\partial t$ and $\nu \nabla^2 \mathbf{u}$ terms in (3). Considering the terms that remain, we get

$$-2\Omega v_I = -\frac{1}{\rho} \frac{\partial p_I}{\partial x} \quad (5)$$

$$2\Omega u_I = -\frac{1}{\rho} \frac{\partial p_I}{\partial y} \quad (6)$$

$$0 = -\frac{1}{\rho} \frac{\partial p_I}{\partial z} \quad (7)$$

from the x -, y - and z -components of (3) respectively, and we get

$$\frac{\partial u_I}{\partial x} + \frac{\partial v_I}{\partial y} + \frac{\partial w_I}{\partial z} = 0 \quad (8)$$

from the continuity equation.

Now here's some interesting reasoning. From (7), p_I is independent of z . This means, then, that $\partial p_I/\partial x$ and $\partial p_I/\partial y$ are also independent of z , so from (5) and (6),

$$u_I \text{ and } v_I \text{ are independent of } z.$$

Also from (5) and (6), we find

$$\begin{aligned} \frac{\partial v_I}{\partial y} &= \frac{\partial}{\partial y} \left(\frac{1}{2\Omega\rho} \frac{\partial p_I}{\partial x} \right) = \frac{1}{2\Omega\rho} \frac{\partial^2 p_I}{\partial y \partial x} \\ \frac{\partial u_I}{\partial x} &= -\frac{\partial}{\partial x} \left(\frac{1}{2\Omega\rho} \frac{\partial p_I}{\partial y} \right) = -\frac{1}{2\Omega\rho} \frac{\partial^2 p_I}{\partial x \partial y} \end{aligned}$$

and since the order of partial differentiation can be exchanged,

$$\frac{\partial v_I}{\partial y} = -\frac{\partial u_I}{\partial x}.$$

Plugging this into (8), we get

$$\frac{\partial w_I}{\partial z} = 0,$$

which means that w_I is independent of z . Combining this with the result for u_I and v_I above, we have that \mathbf{u}_I is independent of z . This is known as the *Taylor-Proudman theorem*, and can be stated:

- For a system in uniform rotation with small velocities \mathbf{u} relative to the rotating coordinate frame, the steady solution \mathbf{u}_I of the inviscid equations of motion will not vary along the axis of rotation.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 27 - Boundary layers: Ekman boundary layers

Reading: Acheson, §8.5-8.6.

In the last lecture we began to consider the problem of the flow between parallel boundaries (at $z = 0$ and $z = L$) that are rotating steadily about the z -axis at slightly different angular velocities. We were able to determine that the velocity of the inviscid, main flow is independent of z (a result known as the *Taylor-Proudman theorem*), which holds generally for coordinate systems in uniform rotation where the fluid velocity relative to those coordinates is small. Now let's consider our particular geometry, taking into account the effect of the walls.

Assuming that the Reynolds number defined by

$$Re = \frac{\Omega L^2}{\nu}$$

is large, we expect that the flow will be inviscid in the region away from the boundaries, and will have thin viscous layers on the walls. Let's focus on the $z = 0$ boundary. The equations for the velocity field (valid for small departures from uniform rotation at angular velocity $\vec{\Omega}$) were

$$\frac{\partial \mathbf{u}}{\partial t} + 2\vec{\Omega} \times \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} \quad (1)$$

$$\nabla \cdot \mathbf{u} = 0. \quad (2)$$

We will apply our usual boundary layer assumption, that the velocity components vary much more rapidly in the direction normal to the boundary than in the other directions. In this case that means that the z -derivatives of velocity are much larger than the x - or y -derivatives, so (1) becomes (noting also that the flow is steady)

$$2\vec{\Omega} \times \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \frac{\partial^2 \mathbf{u}}{\partial z^2}, \quad (3)$$

but the incompressibility condition remains $\nabla \cdot \mathbf{u} = 0$, using the same reasoning we used in deriving the 2-D boundary layer equations before.

The components of (3) work out as

$$\begin{aligned} -2\Omega v &= -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \frac{\partial^2 u}{\partial z^2} \\ 2\Omega u &= -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \frac{\partial^2 v}{\partial z^2} \\ 0 &= -\frac{1}{\rho} \frac{\partial p}{\partial z} + \nu \frac{\partial^2 w}{\partial z^2}. \end{aligned} \quad (4)$$

and the continuity equation written out is

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0. \quad (5)$$

From (5), we conclude that

$$|w| \ll |u|, |v|$$

since all the terms in (5) need to be of the same order, and the x - and y -derivative components are much smaller than the z -component. Then, we can also conclude (from 4) that

$$\left| \frac{\partial p}{\partial z} \right| \ll \left| \frac{\partial p}{\partial x} \right|, \left| \frac{\partial p}{\partial y} \right|$$

meaning

$$p \approx p(x, y). \quad (6)$$

Again, analogously to the reasoning we used with the 2-D boundary layers, (6) means that the pressure field in the boundary layers is basically the same as the pressure in the inviscid, main flow. From the last lecture, we have the result

$$\begin{aligned} -2\Omega v_I &= -\frac{1}{\rho} \frac{\partial p}{\partial x} \\ 2\Omega u_I &= -\frac{1}{\rho} \frac{\partial p}{\partial y} \end{aligned}$$

where the subscript I denotes the velocity component values in the inviscid main flow, and we've removed the subscript from p because the pressure field in the inviscid main flow is also the pressure field in the boundary layers. Putting this result into (4), we get

$$\begin{aligned} -2\Omega(v - v_I) &= \nu \frac{\partial^2 u}{\partial z^2} \\ 2\Omega(u - u_I) &= \nu \frac{\partial^2 v}{\partial z^2}. \end{aligned} \quad (7)$$

The book works through the solution of these equations (p. 282), which gives

$$\begin{aligned} u &= u_I - e^{-z_*} (u_I \cos z_* + v_I \sin z_*) \\ v &= v_I - e^{-z_*} (v_I \cos z_* - u_I \sin z_*), \end{aligned} \quad (8)$$

($z_* = \sqrt{\Omega/\nu} z$) which is known as the *Ekman boundary layer*. It turns out that just at the outside of this boundary layer ($z_* \rightarrow \infty$), the z -component of velocity is important. First write

$$\left(\frac{\Omega}{\nu} \right)^{1/2} \frac{\partial w}{\partial z_*} = \frac{\partial w}{\partial z},$$

using the definition of z_* , then note

$$\begin{aligned} \left(\frac{\Omega}{\nu} \right)^{1/2} \frac{\partial w}{\partial z_*} &= \frac{\partial w}{\partial z} = - \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) \\ &= \left(\frac{\partial v_I}{\partial x} + \frac{\partial u_I}{\partial y} \right) e^{-z_*} \sin z_* - \left(\frac{\partial u_I}{\partial x} + \frac{\partial v_I}{\partial y} \right) (1 - e^{-z_*} \cos z_*) \\ &= \left(\frac{\partial v_I}{\partial x} + \frac{\partial u_I}{\partial y} \right) e^{-z_*} \sin z_*, \end{aligned} \quad (9)$$

where the last step is true because $\partial u_I/\partial x + \partial v_I/\partial y = 0$ from the last lecture.

We can integrate (9) from $z_* = 0$, where $w = 0$, to $z_* = \infty$ to get $w(z_* = \infty)$, the z -component of the velocity at the edge of the Ekman boundary layer. The result is

$$w_E(x, y) = \frac{1}{2} \left(\frac{\nu}{\Omega} \right)^{1/2} \left(\frac{\partial v_I}{\partial x} - \frac{\partial u_I}{\partial y} \right),$$

but the term in parentheses is just the z -component of $\nabla \times \mathbf{u}_I$, the vorticity in the inviscid main flow. We can denote this vorticity component as ω_I , so

$$w_E(x, y) = \frac{1}{2} \left(\frac{\nu}{\Omega} \right)^{1/2} \omega_I, \quad (10)$$

Finally, what happens if the boundaries are rotating at different angular velocities, as with our problem? If we let Ω_B be the angular velocity of the bottom boundary relative to the rotating coordinate system, and Ω_T be the angular velocity of the top boundary relative to the rotating coordinate system, we get

$$w_E(x, y) = \left(\frac{\nu}{\Omega} \right)^{1/2} \left(\frac{1}{2} \omega_I - \Omega_B \right) \quad (11)$$

just outside the bottom Ekman layer, and

$$w_E(x, y) = \left(\frac{\nu}{\Omega} \right)^{1/2} \left(\Omega_T - \frac{1}{2} \omega_I \right) \quad (12)$$

just outside the top Ekman layer.

Now let's focus back on the inviscid main flow. The velocity components given by (11) and (12) have to match the inviscid main flow just outside the bottom and top boundary layers, respectively. But we saw in the last lecture that the inviscid main flow velocity is independent of z , which means that (11) and (12) have to match each other. This gives

$$\omega_I = \Omega_B + \Omega_T.$$

For our problem (Acheson's Fig. 8.10) the coordinate system and the bottom boundary have angular velocity Ω , and the top boundary has angular velocity $\Omega(1 + \epsilon)$, so $\Omega_B = 0$ and $\Omega_T = \Omega\epsilon$. Thus

$$\omega_I = \Omega\epsilon. \quad (13)$$

But remember from our discussion of the Navier-Stokes equations a while back that the vorticity is equal to double the local angular velocity. Thus, (13) tells us that the inviscid main flow between our rotating boundaries rotates at an angular velocity

$$\frac{1}{2} \Omega\epsilon,$$

that is, the main flow has angular velocity that is halfway between the angular velocities of the two boundaries.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 28 - Waves: the basics

Reading: Acheson, §3.1-3.3.

We're going to go back in the book briefly to touch on the subject of waves. The topic of waves is very extensive and covers surface waves, interfacial waves, sound or pressure waves, shocks, solitons and a variety of other phenomena that fill textbooks all by themselves. Our goal is just to cover some fundamental ideas about waves, both to get a feel for the subject, and to learn enough to look at the area of flow instability.

We will consider a simple harmonic surface wave, in which the position of the surface is defined by

$$y = \eta(x, t) = A \cos(kx - \omega t) = \eta(\theta), \quad (1)$$

(A is a constant), where we define

$$\theta(x, t) = kx - \omega t.$$

The driving force behind the wave motion will be gravity. Sinusoidal waves are interesting because Fourier theory tells us that we can decompose any function into sinusoidal components of different wavelengths. So individual waves defined by (1) are the building blocks of general waves and disturbances.

To determine the speed, c , of the wave, we plant ourselves on one particular point on the wave, i.e. fixed θ . From time t to $t + \Delta t$, that point on the wave will have moved from spatial position x to $x + \Delta x$, where x , t , Δx and Δt are related by

$$\theta(x, t) = \theta(x + \Delta x, t + \Delta t) = k(x + \Delta x) - \omega(t + \Delta t).$$

The speed of the wave is then given by

$$c = \frac{\Delta x}{\Delta t} = \frac{\omega}{k}.$$

As we will see later, ω is given by

$$\omega = \sqrt{gk}, \quad (2)$$

where g is the gravitational acceleration, so

$$c = \sqrt{\frac{g}{k}}. \quad (3)$$

This tells us that waves with different k travel at different speeds.

To illustrate the meaning of k , we will determine the wavelength, λ , of the wave defined by (1), where λ is defined as the spatial distance between successive peaks at a given time. Recognizing that in terms of θ , successive peaks are separated by $\Delta\theta = 2\pi$, we can write

$$\theta(x, t) + 2\pi = \theta(x + \lambda, t),$$

which gives us

$$kx - \omega t + 2\pi = k(x + \lambda) - \omega t.$$

Thus the wavelength of the wave can be written

$$\lambda = \frac{2\pi}{k}.$$

(We can follow a similar analysis to find the period, T , of the wave, defined as the time between successive peaks as seen by an observer at a fixed location.) Going back to (3), we see that waves of longer wavelength (smaller k) travel faster. The property whereby individual waves of different wavelengths travel at different speeds is known as *dispersion*.

Dispersion explains an interesting property of groups of waves. By ‘group of waves’ (some authors use the term ‘wave packet’) we mean a pattern of waves that occupies a finite area in space, not bounded by walls. Acheson points out that while an observer viewing a group of waves, which consists of a multitude of different Fourier components (each defined by (1) with its own k and ω), may count N peaks in that group at any particular time, that observer would count more than N peaks as the group passed a fixed location. This is because the velocity of the group of waves is different from the velocity of the individual peaks. The speed of the whole group of waves is given by

$$c_g = \frac{d\omega}{dk},$$

which, using (2), becomes

$$c_g = \frac{1}{2} \sqrt{\frac{g}{k}} = \frac{1}{2} c,$$

using (3). So, the group of waves travels half as fast as the individual wave peaks themselves.

1 Surface waves on deep water

While there’s a lot of interesting background material on waves in §3.1 in the text, we’re going to skip the stuff on sound waves and solitary waves and go directly to the example of surface waves on deep water. We will treat the water as an ideal fluid (negligible viscous forces). We are interested in two-dimensional surface waves, so the velocity field is

$$\mathbf{u} = [u(x, y, t), v(x, y, t), 0],$$

and we will assume that the flow is irrotational, so

$$\omega_z = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}. \quad (4)$$

The irrotationality of the flow is ensured if the fluid is initially at rest, i.e. initially has no vorticity; we saw all the way back in the first chapter that in two-dimensional flow of ideal fluids subject to gravity, the vorticity of each fluid element is conserved. Because the fluid is irrotational, there exists a velocity potential, $\phi(x, y, t)$, such that

$$u = \frac{\partial \phi}{\partial x}, \text{ and } v = \frac{\partial \phi}{\partial y}.$$

Also, because the fluid is incompressible, we have

$$\nabla \cdot \mathbf{u} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = \frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} = \nabla^2 \phi = 0. \quad (5)$$

The position of the free surface will be given by

$$y = \eta(x, t),$$

and it is the deformation of this surface that will generate the fluid motion we're interested in.

We will require that fluid particles on the surface remain there (this implicitly eliminates the possibility of large disturbances, for example). If we define

$$F(x, y, t) = y - \eta(x, t),$$

then the free surface is defined as the surface where $F = 0$. Since fluid elements on the free surface have to stay there, that means F is constant (and = 0) for those fluid elements, so in particular we can write

$$\frac{DF}{Dt} = \frac{\partial F}{\partial t} + (\mathbf{u} \cdot \nabla)F = 0 \quad \text{on the surface.} \quad (6)$$

Using the following relations,

$$\begin{aligned} \frac{\partial F}{\partial t} &= -\frac{\partial \eta}{\partial t} \\ u \frac{\partial F}{\partial x} &= -u \frac{\partial \eta}{\partial x} \\ v \frac{\partial F}{\partial y} &= v, \end{aligned}$$

(6) becomes

$$\frac{\partial \eta}{\partial t} + u \frac{\partial \eta}{\partial x} = v \quad \text{on the surface.} \quad (7)$$

1.1 Small-amplitude approximations

Now, we'll impose the requirement that $\eta(x, t)$, the free surface displacement, and the velocity components u and v and their derivatives be 'small'. (The 'smallness' will amount to requiring that the surface displacement be $\ll \lambda$, the wavelength of the disturbances.) Then, in (7), we will throw away terms that are the products of small quantities (this is called *linearizing*), leaving us with

$$v(x, \eta, t) = \frac{\partial \eta}{\partial t}, \quad (8)$$

where the arguments for v establish that we're talking about v at the surface. Since η is small, we can expand $v(x, \eta, t)$ around $\eta = 0$ in a Taylor series. In general, for a function $f(x)$, the Taylor expansion around $x = 0$ is

$$f(x) = f(0) + x \frac{df}{dx}(0) + \frac{x^2}{2!} \frac{d^2 f}{dx^2}(0) + \frac{x^3}{3!} \frac{d^3 f}{dx^3}(0) + \dots$$

For our problem, $v(x, \eta, t)$ expands as

$$v(x, \eta, t) = v(x, 0, t) + \eta \frac{\partial v}{\partial y}(x, 0, t) + \dots,$$

and once again we toss out anything involving products of small quantities, so (8) becomes

$$v(x, 0, t) = \frac{\partial \phi}{\partial y}(x, 0, t) = \frac{\partial \eta}{\partial t}$$

using $v = \partial \phi / \partial y$, where we can impose the condition $y = 0$ instead of $y = \eta$ because of the linearization.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 29 - Waves: more basics

Reading: Acheson, §3.1-3.3.

In the last lecture, we started to consider the problem of two-dimensional surface waves on deep water, for which the fluid motion is driven by gravity. We defined the free-surface position as

$$y = \eta(x, t),$$

with the implicit assumption that $y = 0$ is the position of the *undisturbed* free surface. Thus $\eta(x, t)$ can be interpreted as the free surface displacement due to the wave motion; we will also refer to $\eta(x, t)$ as to ‘disturbances.’ The following relation pertains between the components of the fluid velocity, u and v , on the surface, and the free-surface position $y = \eta(x, t)$:

$$\frac{\partial \eta}{\partial t} + u \frac{\partial \eta}{\partial x} = v \quad \text{on the surface.} \quad (1)$$

Imposing the condition that the disturbances, η , the velocity components, u and v , and their derivatives be small, we linearized (1) to get

$$v(x, 0, t) = \frac{\partial \phi}{\partial y}(x, 0, t) = \frac{\partial \eta}{\partial t}, \quad (2)$$

(ϕ is the velocity potential) which says that the ‘speed’ at which the free surface moves in the y -direction is equal to the v -component of the fluid velocity at $y = 0$.

1 Bernoulli’s equation for unsteady irrotational flow

We can get another free-surface condition relating the disturbances η to the velocity field by looking at the following form of Euler’s equation for the velocity of an ideal fluid –

$$\frac{\partial \mathbf{u}}{\partial t} + (\nabla \times \mathbf{u}) \times \mathbf{u} = -\nabla \left(\frac{p}{\rho} + \frac{1}{2} |\mathbf{u}|^2 + \chi \right), \quad (3)$$

where χ is the velocity potential. (Recall that $\mathbf{g} = -\nabla \chi$, and since $\mathbf{g} = -g\hat{e}_y$ for the present problem, we have $\chi = gy$.) since the flow in this problem is irrotational, we have also

$$\begin{aligned} \boldsymbol{\omega} &= \nabla \times \mathbf{u} = 0 \\ \mathbf{u} &= \nabla \phi \end{aligned}$$

which, inserted in (3), gives

$$\frac{\partial}{\partial t} \nabla \phi = -\nabla \left(\frac{p}{\rho} + \frac{1}{2} |\mathbf{u}|^2 + \chi \right),$$

which can be written, interchanging the spatial and time derivative on the left side,

$$\nabla \left(\frac{\partial \phi}{\partial t} \right) = -\nabla \left(\frac{p}{\rho} + \frac{1}{2} |\mathbf{u}|^2 + \chi \right). \quad (4)$$

We can get rid of the gradient operator ∇ in (4) by integrating. Remember that the gradient can be interpreted

$$\nabla \phi \cdot d\mathbf{s} = \frac{d\phi}{ds} ds = d\phi_s$$

where $d\mathbf{s}$ is an arbitrary vector with infinitesimal magnitude ds . To integrate (4), then, we first take the dot product of both sides with some vector $d\mathbf{s}$. The integral of both sides can then be written

$$d\left(\frac{\partial\phi}{\partial t}\right) = -d\left(\frac{p}{\rho} + \frac{1}{2}|\mathbf{u}|^2 + \chi\right), \quad (5)$$

which gives

$$\frac{\partial\phi}{\partial t} = -\left(\frac{p}{\rho} + \frac{1}{2}|\mathbf{u}|^2 + \chi\right) + G(t), \quad (6)$$

where the arbitrary ‘constant’ of integration G is allowed to be a function of t because the integration is over spatial variables only. Rearranging this, we get

$$\frac{\partial\phi}{\partial t} + \frac{p}{\rho} + \frac{1}{2}|\mathbf{u}|^2 + \chi = G(t), \quad (7)$$

which is *Bernoulli’s equation for unsteady, irrotational flow*. By (7), the quantity

$$\frac{\partial\phi}{\partial t} + \frac{p}{\rho} + \frac{1}{2}|\mathbf{u}|^2 + \chi \quad (8)$$

is constant in space.

Now let’s look specifically at our problem. At the surface, $y = \eta(x, t)$, the pressure is equal to the atmospheric pressure, p_0 , which can be treated as constant. Then (7) gives us

$$\frac{\partial\phi}{\partial t} + \frac{1}{2}(u^2 + v^2) + g\eta = G(t), \quad (9)$$

where we’ve incorporated the pressure term into G , and used $\chi = gy$. Now, we also note that G is arbitrary, because we can write the velocity potential as

$$\phi = \phi_s(x, y, t) + \phi_t(t), \quad (10)$$

where now $\phi_t(t)$ is arbitrary, because we are only interested in the velocity potential through the expression

$$\mathbf{u} = \nabla\phi = \nabla\phi_s, \quad (11)$$

and $\phi_t(t)$ thus has no effect on the resulting \mathbf{u} . In (9), then, we are free to choose $G(t) = 0$, giving us

$$\frac{\partial\phi}{\partial t} + \frac{1}{2}(u^2 + v^2) + g\eta = 0. \quad (12)$$

We can now apply our small-amplitude approximations to this. Specifically, we are treating the disturbances, η , and the velocity components, u and v , as small. The condition that u and v are small means that the spatial derivatives of ϕ are small, and we assume here that we can extend this to say that the time derivative of ϕ is also small. Linearizing (12) gives us

$$\frac{\partial\phi}{\partial t}(x, 0, t) + g\eta(x, t) = 0, \quad (13)$$

where this condition is imposed at $y = 0$ rather than $y = \eta$ by the same arguments we used in the last lecture to derive (2).

2 Dispersion relation

We are going to look for a solution to (2) and (13) of the form

$$\eta(x, t) = A \cos(kx - \omega t). \quad (14)$$

Look at (13) first. Plugging in (14), we get

$$\frac{\partial \phi}{\partial t}(x, 0, t) = -gA \cos(kx - \omega t),$$

and integrating over t , we get

$$\phi(x, 0, t) = \frac{gA}{\omega} \sin(kx - \omega t).$$

We have to be a little careful in interpreting this because ϕ in this expression is explicitly evaluated at a fixed y . What we can hypothesize is that ϕ has the form

$$\phi(x, y, t) = f(y) \sin(kx - \omega t). \quad (15)$$

We can make the same hypothesis by considering (2).

The potential ϕ has to satisfy Laplace's equation, i.e.

$$\nabla^2 \phi = \frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2},$$

so

$$-f(y)k^2 \sin(kx - \omega t) + f''(y) \sin(kx - \omega t) = 0$$

which gives

$$f'' - k^2 f = 0.$$

The general solution of this is (as seen in any textbook on differential equations)

$$f = Ce^{ky} + De^{-ky}. \quad (16)$$

We can simplify this by considering what happens at $y \rightarrow -\infty$ (remember that we're considering the problem of surface waves on deep water). Look at the u -component of velocity, which is (using (15))

$$u = \frac{\partial \phi}{\partial x} = f(y)k \cos(kx - \omega t) = (Ce^{ky} + De^{-ky})k \cos(kx - \omega t).$$

In order for this to be finite as $y \rightarrow -\infty$, we must have $D = 0$. Thus (16) becomes

$$f = Ce^{ky},$$

and we can write (15) as

$$\phi(x, y, t) = Ce^{ky} \sin(kx - \omega t). \quad (17)$$

Now we want to get some idea of how the constants in (17) relate to each other. Inserting (17) into (2), we get

$$Cke^{k(0)} \sin(kx - \omega t) = A\omega \sin(kx - \omega t)$$

or

$$Ck = A\omega, \quad (18)$$

and inserting (17) into (13) we get

$$-Ce^{k(0)}\omega \cos(kx - \omega t) + gA \cos(kx - \omega t) = 0$$

or

$$-C\omega + gA = 0. \quad (19)$$

Equation (18) tells us $C = A\omega/k$, so

$$\phi(x, y, t) = \frac{A\omega}{k} e^{ky} \sin(kx - \omega t). \quad (20)$$

Also, (18) and (19) together give us

$$\omega^2 = gk \quad (21)$$

which we anticipated earlier. This is the *dispersion relation* that tells us how ω and k relate, i.e. (remembering that the wave speed is given by $c = \omega/k$ and the wavelength is $2\pi/k$) how the wave speed varies with the wavelength.

3 Particle paths

From our result (20) for the velocity potential, the velocity components are

$$\begin{aligned} u &= A\omega e^{ky} \cos(kx - \omega t) \\ v &= A\omega e^{ky} \sin(kx - \omega t). \end{aligned}$$

We will assume that the instantaneous position of any particle, which we'll denote by (x', y') , is very close to its average position, (\bar{x}, \bar{y}) . We can then write

$$\begin{aligned} u &= \frac{\partial x'}{\partial t} \approx A\omega e^{k\bar{y}} \cos(k\bar{x} - \omega t) \\ v &= \frac{\partial y'}{\partial t} \approx A\omega e^{k\bar{y}} \sin(k\bar{x} - \omega t) \end{aligned}$$

which integrates to

$$\begin{aligned} x' &= -Ae^{k\bar{y}} \sin(k\bar{x} - \omega t) \\ y' &= Ae^{k\bar{y}} \cos(k\bar{x} - \omega t) \end{aligned}$$

These trajectories describe circles (Fig. 1).

Two observations. The radii of the trajectories is $Ae^{k\bar{y}}$, which decreases exponentially as we go deeper (more negative y), just like the fluid velocities do. The other observation is that near the peak of a disturbance, there is forward motion of the water, and near the troughs, there is rearward motion; you can see this very readily by watching a floating object (say, yourself) as it encounters a series of wavefronts.

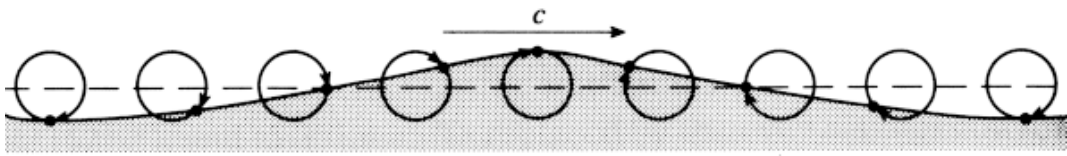


Figure 1: Particle paths (figure from Faber, *Fluid dynamics for physicists*).

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 30 - Instability: introduction

Reading: Acheson, §9.1-9.2.

The subject of flow instability (also referred to as hydrodynamic instability or laminar flow instability) is our bridge to one of the most interesting problems, if not the most interesting problem, in all of the physical sciences; that problem being *turbulence*.

The classic experiment in turbulence is that of Osborne Reynolds that Acheson describes in §9.1. Reynolds's experiment concerned flow in a tube of circular cross section. To mark the flow pattern, Reynolds used thin streams of dye introduced into the flow in the tube. Reynolds's observations seem simple; for low velocities the stream of dye ran down the tube basically undisturbed (emblematic of laminar flow), while for higher velocities, there would appear a point in the tube where the dye pattern suddenly erupted into seeming randomness and disorder (emblematic of turbulence). To quantify the point of transition between laminar and turbulent flow, Reynolds used (logically enough) the Reynolds number:

$$Re = \frac{UL}{\nu}$$

where U is the characteristic velocity of the flow, L is a characteristic length scale, and ν is the kinematic viscosity of the fluid.

There are numerous striking aspects about the transition from laminar to turbulent flow. A couple of them are:

- As Reynolds observed (and as anyone can see by looking at the plume of smoke rising from a cigarette), the transition from laminar flow to turbulence is very abrupt, occurring at a particular value of the Reynolds number (the *critical* Reynolds number, Re_c). Flows are either laminar or turbulent, they can't be slightly laminar or kind of turbulent. This can be thought of as a symptom of the *nonlinearity* of the Navier-Stokes equations and of turbulence. Think of the inputs to a physical system as lying on the x -axis of a plot and of the outputs as being on the y -axis, and for fluid flow systems consider the Reynolds number as quantifying the input and consider the amount of disorder as the output. If fluid flows were linear, then, say, doubling the input should double the output. However this is clearly not true when the increase in Reynolds number goes across the critical value Re_c .
- Maybe more interesting, there is really no reason why we might predict the existence of turbulence. Think of our solutions for viscous flow in a pipe or between parallel plates. There was nothing in our solutions for those flows that indicated that they should only be valid for low Reynolds numbers only.

So where does turbulence come from? Basically, turbulence can be thought to arise because the physical world isn't ideal. Walls are never completely still or smooth, flow profiles are never completely uniform, boundary conditions are never completely steady (in much the same way that friction is never zero, and vacuums are never completely free of particles, and so on). If walls were completely steady, etc., then we *could* just bump up the flow velocity in a tube without changing the flow qualitatively. But because there are always irregularities, however small, we have to worry about what happens to those irregularities as we change our flow parameters. In the case of Reynolds's experiment, when he increased the Reynolds number, the infinitesimal irregularities in his wall smoothness, etc. eventually grew until they made the drastic qualitative change in his flow that he observed as turbulence. To treat this phenomenon mathematically, we enter the area of stability (or instability) theory.

1 Linear stability theory and Kelvin-Helmholtz instability

We will consider the two-dimensional flow system in which one deep layer of inviscid fluid, with density ρ_2 , flows at uniform speed U over another deep layer of inviscid fluid, with density $\rho_1 > \rho_2$, which is at rest (Fig. 1). We permit there to be surface tension on the interface between the fluids. Now suppose that the interface (given by $y = \eta(x, t)$) between the two fluids is subject to a

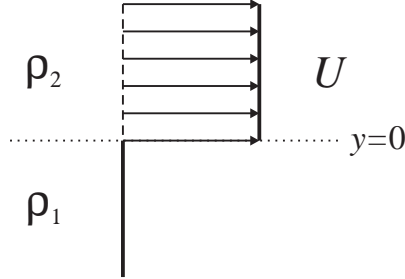


Figure 1: Flow system illustrating Kelvin-Helmholtz instability.

sinusoidal, travelling-wave disturbance, so that

$$\eta(x, t) = A \operatorname{Re} \left[e^{i(kx - \omega t)} \right] \quad (1)$$

where we recall that $e^{i\theta} = \cos \theta + i \sin \theta$, and the ‘Re’ indicates that we are taking the real part of the complex exponential.

We are interested in the *temporal stability* problem for this flow system (as opposed to the *spatial stability* problem, more on that later). That is, given an arbitrary initial disturbance in space, how does that disturbance evolve in time? Applying Fourier theory, we can express a general disturbance of the interface between our two fluids as

$$\eta(x, t) = \int_{-\infty}^{\infty} a(k) \operatorname{Re} \left[e^{i(kx - \omega t)} \right] dk. \quad (2)$$

Written in this form, a general disturbance is made up of individual disturbances, each of the form (1), with different values of the wavenumber, k (recall $2\pi/k = \lambda$, where λ is the wavelength). We will make the following definition:

- A flow system is called *stable* when subject to a general disturbance of the form (2) if the magnitude of the disturbance remains finite for all time. This requires that the magnitudes of individual disturbances of the form (1) remain finite for all time, for all values of k .

Our next question is, how do we determine the magnitude of a disturbance as a function of time?

The key will be to find the dispersion relation, $\omega = \omega(k)$. For the temporal instability problem, we will require that the wavenumber k be real, but the *angular frequency*, ω , will in general be a complex number, which we’ll write as

$$\omega = \omega_R + i\omega_I. \quad (3)$$

What is the significance of the complex angular frequency? If we plug (3) into (1), we get

$$\begin{aligned} \eta(x, t) &= A \operatorname{Re} \left[e^{i(kx - \omega_R t - i\omega_I t)} \right] \\ &= A \operatorname{Re} \left[e^{\omega_I t} e^{i(kx - \omega_R t)} \right]. \end{aligned} \quad (4)$$

The magnitude of the imaginary part of the exponential is always bounded by 1, because it's just sines and cosines. However, the term $e^{\omega_I t}$ is a different story. If $\omega_I > 0$, then this term clearly grows exponentially as time evolves. So our stability definition from above can be recast in the following form:

- A system is temporally stable if $\omega_I \leq 0$ for all values of k .

Also, and this is a significant point, we will consider infinitesimal disturbances, so that we can linearize the necessary equations, as we did with the wave problem of the past couple of lectures; with this condition the procedure we're describing is called *linear stability theory*.

Now let's turn to the particular flow system in Fig. 1. In the homework you will show that, with the disturbances treated as small and linearizing the appropriate equations, the following relation holds:

$$\overbrace{(\rho_1 + \rho_2)}^a \omega^2 \overbrace{-2\rho_2 U k}^b \omega + \overbrace{\rho_2 U^2 k^2 - |k|(k^2 T + (\rho_1 - \rho_2)g)}^c = 0, \quad (5)$$

which is a quadratic equation in the variable ω . The roots (solutions) to (5), of course, have the form

$$\omega = \frac{-b \pm \sqrt{b^2 - 4ac}}{2a} = \omega(k), \quad (6)$$

where the form $\omega(k)$ expresses that the solutions to (5) represent the dispersion relation for the system.

From (6), it is readily seen that (a) there will be an imaginary component to ω if $b^2 - 4ac < 0$, and that (b) in that case, one of the roots of (5) will have imaginary part greater than zero, and the other will have imaginary part less than zero. So for the system to be temporally stable, the solutions to (5) *cannot* have an imaginary part, i.e. we must have $b^2 - 4ac \geq 0$. For this problem, this condition is written

$$b^2 - 4ac = 4\rho_2^2 U^2 k^2 - 4(\rho_1 + \rho_2)[\rho_2 U^2 k^2 - |k|(k^2 T + (\rho_1 - \rho_2)g)] \geq 0. \quad (7)$$

Playing with the algebra, we have

$$4\rho_2 U^2 k^2 [-\rho_1] \geq 4(\rho_1 + \rho_2) \overbrace{|k|(k^2 T + (\rho_1 - \rho_2)g)}^{\text{"D"}} \\ U^2 \leq \left(|k|T + (\rho_1 - \rho_2) \frac{g}{|k|} \right) \left(\frac{1}{\rho_1} + \frac{1}{\rho_2} \right). \quad (8)$$

So, we know that a disturbance η , given by (1), with wavenumber k , is stable if (8) holds. But for the flow system to be stable when subject to a general disturbance (2), we need this condition (8) to hold for all values of k .

To ensure this, we just need to make sure that (8) holds for the particular value of k that minimizes the quantity D . To find that k , first take the derivative of D with respect to $|k|$, then set that equal to zero –

$$\frac{d}{d|k|} D = \frac{d}{d|k|} \left(|k|T + (\rho_1 - \rho_2) \frac{g}{|k|} \right) = \left(T - (\rho_1 - \rho_2) \frac{g}{|k|^2} \right) = 0,$$

which holds when $|k|$ satisfies

$$|k| = \sqrt{(\rho_1 - \rho_2) \frac{g}{T}}.$$

Then, D is given by

$$D = 2\sqrt{(\rho_1 - \rho_2)gT}, \quad (9)$$

and, putting this back into (8), we find that the system in Fig. 1 is temporally stable if

$$U^2 \leq 2\sqrt{(\rho_1 - \rho_2)gT} \left(\frac{1}{\rho_1} + \frac{1}{\rho_2} \right). \quad (10)$$

Some observations; from (10), the system becomes more stable (i.e. the critical value of velocity U_c below which the system is stable increases) if the density difference $\rho_1 - \rho_2$ between the fluids increases, or if the surface tension T on the interface increases. Also, gravity is a stabilizing influence.

(This treatment dealt with temporal stability, which is sometimes also called absolute stability. The problem of spatial stability involves following a particular wave packet as it evolves in space. In that case, we would allow our disturbances (1) to have real values of the angular velocity, ω , but complex values of wavenumber, k . Spatial stability is also called convective stability.)

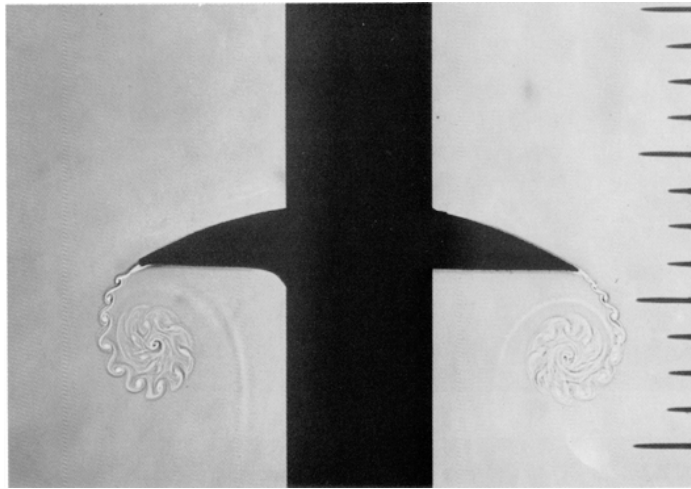


Figure 2: An interesting example of Kelvin-Helmholtz-like instability (from van Dyke, *An Album of Fluid Motion*). A 3 inch square, horizontal plate starts at rest and is suddenly accelerated upward. The photo is taken 6.5 milliseconds after the start of the motion, when the speed has just reached the maximum 24 ft/s. The pins on the right are spaced 1/4 inch apart; the lowermost pin represents the starting position of the plate.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 31 - Instability: more on Kelvin-Helmholtz, and thermal convection

Reading: Acheson, §9.2-9.3.

To recap what we did with Kelvin-Helmholtz instability in the last lecture –

- We defined our undisturbed, two-dimensional flow system: a deep layer of inviscid fluid, density ρ_2 , for $y > 0$, and a deep layer of inviscid fluid of density ρ_1 for $y < 0$, with $\rho_1 > \rho_2$. The interface is permitted to have surface tension. Initially, the upper fluid moves at uniform speed U in the x -direction and the lower fluid is at rest (Fig. 1).

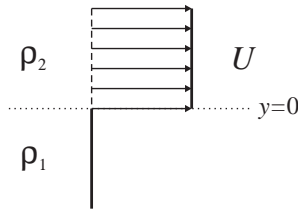


Figure 1: Undisturbed flow for the Kelvin-Helmholtz problem.

- We allowed the interface to be subject to sinusoidal, traveling-wave disturbances with infinitesimal magnitude. We then wanted to see how the initially, spatially-defined disturbances evolved with time; specifically, do they increase or decrease in magnitude with time? We call a flow system stable to a particular disturbance if that disturbance dies away in magnitude after it's introduced. For this *temporal stability* analysis, we considered disturbances with real wavenumbers k but complex angular frequencies ω .
- We showed that disturbances grow exponentially with time, and thus destroy the undisturbed flow pattern (Fig. 1), when the imaginary part of the complex angular frequency is greater than zero. What we wanted, then, was to find the dispersion relation for the system, that is, $\omega = \omega(k)$. This would tell us whether a disturbance of a given wavenumber either grows or decays with time.
- Linearizing the appropriate equations, then solving for $\omega(k)$, we were able to show that a given disturbance decays with time (i.e. doesn't grow and eventually destroy the initial flow pattern) when the following relation holds:

$$U^2 \leq \left(|k|T + (\rho_1 - \rho_2) \frac{g}{|k|} \right) \left(\frac{1}{\rho_1} + \frac{1}{\rho_2} \right). \quad (1)$$

- Recognizing that an arbitrary initial disturbance is made up of a combination of sinusoidal disturbances, each with different wavenumber, we said that the flow system of Fig. 1 could be called *temporally stable* if disturbances of all wavenumbers decay with time. Working from (1), we showed that this was true when

$$U^2 \leq 2\sqrt{(\rho_1 - \rho_2)gT} \left(\frac{1}{\rho_1} + \frac{1}{\rho_2} \right). \quad (2)$$

Acheson's Fig. 9.4(b) illustrates this mathematically argument.

1 Thermal convection

Consider a system where viscous fluid sits at rest between two plane rigid boundaries, located at $z = 0$ and $z = d$. We hold the two boundaries at different temperatures T_l and T_u , respectively, where $T_l - T_u = \Delta T > 0$; that is, the lower boundary is hotter than the upper boundary. The fluid nearer the lower boundary will therefore have a lower density, and will, due to buoyancy, have a tendency to rise. Because the system is confined, though, it is not immediately obvious how it will be possible for the fluid near the lower boundary to exchange places with the fluid near the upper boundary. It turns out that the fluid will remain in the initial state of no motion until ΔT exceeds a critical value. We can demonstrate this using linear stability theory.

In this problem the variation of density is significant, so we won't be making our usual incompressible, constant density assumptions. With the Navier-Stokes equations as we've been using them, we have an equation describing mass conservation and one describing the balance of momentum. Here, we will need two more equations: an equation for the (internal) energy, and also an equation of state relating the fluid density to temperature and pressure. To simplify things a little, we'll confine our interest to the case where our viscous fluid is a liquid.

(This problem is not trivial to solve, as you'll see from reading pp. 309–311. We'll concern ourselves mainly with setting up the problem, then discussing the properties of the solution. In setting up the problem, we will basically be linearizing like crazy.)

For liquids, density varies with both temperature and pressure, but the temperature dependence is small and the pressure dependence much smaller still. So we can include in our state equation only a linear temperature dependence, or

$$\rho = \bar{\rho}[1 - \alpha(T - \bar{T})], \quad (3)$$

where \bar{T} is just an arbitrary reference temperature, and $\rho(T = \bar{T}) = \bar{\rho}$. Note: while \bar{T} is arbitrary, (3) only holds if the temperatures we're considering are close to \bar{T} .

For the mass conservation, the general expression we found back in the first homework assignment –

$$\frac{D\rho}{Dt} + \rho \nabla \cdot \mathbf{u} = 0.$$

Because the variations in ρ for liquids are very small, we can basically ignore the $D\rho/Dt$ term, leaving us with

$$\nabla \cdot \mathbf{u} = 0, \quad (4)$$

with which we're quite familiar.

The momentum equation is also familiar:

$$\rho \frac{D\mathbf{u}}{Dt} = -\nabla p + \rho \nu \nabla^2 \mathbf{u} + \rho \mathbf{g}, \quad (5)$$

except we need to recognize now that ρ isn't a constant. We will, however, assume that ν is a constant, independent of temperature.

The energy equation we will use is

$$\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T = \kappa \nabla^2 T, \quad (6)$$

which, as you'll recognize from the last homework, means that temperature (which we think of here equivalently to internal energy) is a conserved scalar quantity. This means that we're neglecting any changes of the total internal energy in the system, e.g. work terms or viscous dissipation.

We want to know first what the temperature distribution is in the undisturbed system where the fluid is at rest. Call this temperature distribution $T_0(z)$. In (6), the terms on the left side are zero, leaving us with

$$0 = \kappa \frac{d^2 T_0}{dz^2}. \quad (7)$$

Integrating twice and applying the boundary conditions, $T(z = d) = T_u$ and $T(z = 0) = T_l$, we get

$$T_0(z) = T_l - \frac{z}{d}(T_l - T_u) = T_l - \frac{z}{d}\Delta T. \quad (8)$$

We can plug (8) into (3) to get the density distribution in the undisturbed fluid –

$$\rho_0(z) = \bar{\rho}[1 - \alpha(T_0(z) - \bar{T})], \quad (9)$$

and using (5), we can find the equation for the pressure distribution in the undisturbed fluid by tossing all the terms with \mathbf{u} in them, leaving us with

$$0 = -\frac{dp_0}{dz} - \rho_0(z)g. \quad (10)$$

At this point, we want to disturb the system, so

$$\begin{aligned} T &= T_0(z) + T_1 \\ \rho &= \rho_0(z) + \rho_1 \\ p &= p_0(z) + p_1 \\ \mathbf{u} &= \mathbf{u}_1 \end{aligned} \quad (11)$$

where the subscript 1 denotes small quantities that are functions of x , y , z and t . Now we plug (11) into our four equations (3), (4), (5) and (6), and linearize.

For the state equation, we get

$$\rho_0 + \rho_1 = \bar{\rho}[1 - \alpha(T_0 + T_1 - \bar{T})],$$

and using (9), we have

$$\rho_1 = -\alpha\bar{\rho}T_1. \quad (12)$$

The continuity equation is simple:

$$\nabla \cdot \mathbf{u}_1 = 0. \quad (13)$$

For the momentum equation, we have

$$(\rho_0 + \rho_1) \left(\frac{\partial \mathbf{u}_1}{\partial t} + (\mathbf{u}_1 \cdot \nabla) \mathbf{u}_1 \right) = -\nabla(p_0 + p_1) + (\rho_0 + \rho_1)\nu \nabla^2 \mathbf{u}_1 + (\rho_0 + \rho_1)\mathbf{g}$$

and throwing out products of small quantities, we leave ourselves with

$$\rho_0 \frac{\partial \mathbf{u}_1}{\partial t} = -\nabla(p_0 + p_1) + \rho_0 \nu \nabla^2 \mathbf{u}_1 + (\rho_0 + \rho_1)\mathbf{g}.$$

Applying (10), we have

$$\rho_0 \frac{\partial \mathbf{u}_1}{\partial t} = -\nabla p_1 + \rho_0 \nu \nabla^2 \mathbf{u}_1 + \rho_0 \mathbf{g}.$$

Finally, we can also linearize (9). Recognizing that $T_0 - \bar{T}$ is a small quantity, we can replace ρ_0 in the above equation with $\bar{\rho}$, giving us

$$\bar{\rho} \frac{\partial \mathbf{u}_1}{\partial t} = -\nabla p_1 + \bar{\rho} \nu \nabla^2 \mathbf{u}_1 + \rho_1 \mathbf{g}. \quad (14)$$

In the energy equation (6), we recognize that T_0 is not a function of time and use (7). This leaves us with

$$\frac{\partial T_1}{\partial t} + \mathbf{u}_1 \cdot \nabla (T_0 + T_1) = \kappa \nabla^2 T_1.$$

Again, we ignore products of small quantities in the second term, and use $\nabla T_0 = dT_0/dz$, giving

$$\frac{\partial T_1}{\partial t} + w_1 \frac{dT_0}{dz} = \kappa \nabla^2 T_1. \quad (15)$$

We'll look at the solutions to the system (12), (13), (14) and (15) in the next lecture.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 32 - Instability: thermal convection, and parallel shear flow

Reading: Acheson, §9.3, 9.5, 9.8.

In the last lecture we considered a viscous fluid at rest between parallel plates at $z = 0$ and $z = d$, where the lower plate is held at temperature T_l and the upper at T_u , and $T_l > T_u$. The equations describing the evolution of small disturbances we found as:

$$\begin{aligned}\rho_1 &= -\alpha\bar{\rho}T_1 \\ \nabla \cdot \mathbf{u}_1 &= 0 \\ \bar{\rho}\frac{\partial \mathbf{u}_1}{\partial t} &= -\nabla p_1 + \bar{\rho}\nu\nabla^2\mathbf{u}_1 + \rho_1\mathbf{g} \\ \frac{\partial T_1}{\partial t} + w_1\frac{dT_0}{dz} &= \kappa\nabla^2T_1.\end{aligned}$$

Courtesy of some manipulation covered in §9.3 of the text (which makes use of differential equation analysis that's a little beyond us), we can first reduce the problem to a problem in the z -component of the velocity disturbance, w_1 . In particular, the form of w_1 is

$$w_1(x, y, t) = W(z)f(x, y)e^{st} \quad (1)$$

where $f(x, y)$ has to satisfy

$$\frac{\partial^2 f}{\partial x^2} + \frac{\partial^2 f}{\partial y^2} + a^2 f = 0, \quad (2)$$

with a being some constant (which can take on *any* value). It is perfectly intuitive that we'd be interested in the z -component of the velocity; when we talk about stability of this system, the issue is whether the buoyancy introduced by the temperature gradient between the boundaries is sufficient to induce large scale motion of the warm fluid upward and the cold fluid downward.

From the form of (1), it is clear that the magnitude of the disturbance w_1 increases with time if $s > 0$. To simplify the problem somewhat Acheson gives the solution for the case where the boundaries are assumed to be stress-free (essentially ignoring viscosity at the boundaries). It turns out that $s > 0$ if the following holds:

$$\frac{\alpha g \Delta T}{\nu \kappa d} > \frac{1}{a^2} \left(a^2 + \frac{N^2 \pi^2}{d^2} \right)^3, \quad (3)$$

where N is any positive integer. In order for us to declare the system to be stable for a particular value of the temperature difference ΔT , (3) needs to hold for all infinitesimal disturbances, i.e. all possible values of N and a . Basically, we want to find the minimum value of the right side of (3). This defines ΔT_c , the critical value of the temperature difference. If $\Delta T < \Delta T_c$, then all infinitesimal disturbances decay with time.

The value of N that minimizes the right side of (3) is obvious. Some simple calculus suffices to find the minimizing value of a . We get

$$N = 1, \text{ and } a_c = \frac{\pi}{\sqrt{2}d}. \quad (4)$$

Defining the Rayleigh number as

$$Ra = \frac{\alpha g \Delta T d^3}{\nu \kappa}, \quad (5)$$

and using (4) in (3), we find that the critical Rayleigh number (above which the system is unstable and below which it's unstable) is

$$Ra_c = \frac{27\pi^4}{4} \approx 658.$$

If we don't impose the stress-free condition on the boundary, Acheson gives the critical Rayleigh number as

$$Ra_c = 1708.$$

Note what it means that the Rayleigh number governs the stability or instability. Increasing the viscosity or thermal conductivity stabilizes the system (raises the value of ΔT before instability sets in), while increasing α (which measures the response of the fluid density to temperature changes) and the spacing of the boundaries destabilizes the system. Acheson discusses the relation of these analytical results to experimental observations in the text.

1 Parallel shear flow: inviscid theory

The last problem in instability we'll consider is that of the stability of parallel shear flow. Reynolds looked at this problem in some detail in the early 1880's, as part of the experimental effort that also involved the tube flow problem we discussed earlier; in the decade or so prior to Reynolds's efforts, Helmholtz, Kelvin and Rayleigh had looked at the problem from a theoretical standpoint. The theoretical work concentrated on the inviscid problem, which is much more tractable analytically than the viscous problem. The viscous analysis really became workable thanks to Orr and Sommerfeld in the early 1900's. We will get to the very influential Orr-Sommerfeld equation for viscous stability of parallel flows in the next lecture.

First, the inviscid case. The problem setup is simple. Suppose we have two plane boundaries at $y = -L$ and $y = L$. We are only interested in two-dimensional flow, so the velocity field has only u - and v -components, and these components vary spatially only in x and y . The instability problem is inherently unsteady, of course, so we have to allow time variation of u and v . The governing equations are the Euler equations:

$$\begin{aligned} \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} &= -\frac{1}{\rho} \frac{\partial p}{\partial x} \\ \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} &= -\frac{1}{\rho} \frac{\partial p}{\partial y} \\ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0. \end{aligned} \tag{6}$$

For our undisturbed flow conditions, we consider the parallel shear flow given by

$$\mathbf{u}_0 = (U(y), 0, 0). \tag{7}$$

Substituting (7) into the Euler equations, it is clear that (7) gives zero left hand side for each of the equations (6). Thus, the parallel shear flow (7) is a solution to (6) for any choice of the velocity profile $U(y)$, provided that the pressure is given by

$$p_0 = \text{const.} \tag{8}$$

There are no boundary conditions on $U(y)$, because the no-slip condition doesn't pertain for inviscid flows.

Now we'll impose two-dimensional disturbances to the undisturbed system. Denoting the (small) disturbances by the subscript 1, we write

$$\begin{aligned}\mathbf{u} &= (U(y) + u_1(x, y, t), v_1(x, y, t), 0) \\ p &= p_0 + p_1(x, y, t).\end{aligned}\tag{9}$$

Plugging (9) into (6) gives us

$$\begin{aligned}\frac{\partial u_1}{\partial t} + (U + u_1) \frac{\partial u_1}{\partial x} + v_1 \frac{d(U + u_1)}{dy} &= -\frac{1}{\rho} \frac{\partial p_1}{\partial x} \\ \frac{\partial v_1}{\partial t} + (U + u_1) \frac{\partial v_1}{\partial x} &= -\frac{1}{\rho} \frac{\partial p_1}{\partial y} \\ \frac{\partial u_1}{\partial x} + \frac{\partial v_1}{\partial y} &= 0\end{aligned}$$

and linearizing, we get

$$\begin{aligned}\frac{\partial u_1}{\partial t} + U \frac{\partial u_1}{\partial x} + v_1 \frac{dU}{dy} &= -\frac{1}{\rho} \frac{\partial p_1}{\partial x} \\ \frac{\partial v_1}{\partial t} + U \frac{\partial v_1}{\partial x} &= -\frac{1}{\rho} \frac{\partial p_1}{\partial y} \\ \frac{\partial u_1}{\partial x} + \frac{\partial v_1}{\partial y} &= 0.\end{aligned}\tag{10}$$

Our experience with surface waves might suggest that we look for solutions for u_1 , v_1 and p_1 that are sinusoidal in form. In fact, we do, except that because the equations (10) have coefficients of the derivatives that are functions of y (in particular, $U = U(y)$), we will look for solutions that have the following form (never mind how we know to do this; it's in the theory of differential equations) –

$$\begin{aligned}u_1 &= \text{Re} \left[\hat{u}(y) e^{i(kx - \omega t)} \right] \\ v_1 &= \text{Re} \left[\hat{v}(y) e^{i(kx - \omega t)} \right] \\ p_1 &= \text{Re} \left[\hat{p}(y) e^{i(kx - \omega t)} \right]\end{aligned}\tag{11}$$

where k is taken to be real, but the ω and the $(\hat{\cdot})$ quantities are complex, similar to what we did in considering the temporal stability of a given form of spatial disturbance:

$$\begin{aligned}\hat{u}, \hat{v}, \hat{p} &\in \mathbf{C}, \\ k &\in \mathfrak{R}, \text{ and } \omega \in \mathbf{C}.\end{aligned}$$

Putting (11) into (10), we get

$$-i(\omega - Uk)\hat{u} + U'\hat{v} = -\frac{1}{\rho} ik\hat{p}\tag{12}$$

$$-i(\omega - Uk)\hat{v} = -\frac{1}{\rho} \hat{p}'\tag{13}$$

$$ik\hat{u} + \hat{v}' = 0\tag{14}$$

where we've taken out the common exponential terms and where $(\cdot)'$ means $d(\cdot)/dy$.

We can manipulate this to get an equation for \hat{v} only. First, rewrite (14) as

$$\hat{u} = -\frac{1}{ik}\hat{v}'$$

and plug this into (12), giving

$$\frac{1}{k}(\omega - Uk)\hat{v}' + U'\hat{v} = -\frac{1}{\rho}ik\hat{p}.$$

Now differentiate this in y , and compare with (13):

$$\overbrace{-U'\hat{v}'}^{\text{cancels}} + \frac{1}{k}(\omega - Uk)\hat{v}'' + U''\hat{v} + \overbrace{U'\hat{v}'}^{\text{cancels}} = -\frac{1}{\rho}ik\hat{p}' = ik[-i(\omega - Uk)] = k(\omega - Uk),$$

giving us

$$\frac{1}{k}(\omega - Uk)\hat{v}'' + U''\hat{v} = k(\omega - Uk).$$

Multiply through by $(\omega - Uk)/k$, and we get

$$\hat{v}'' + \left(\frac{kU''}{\omega - Uk} - k^2 \right) \hat{v} = 0, \quad (15)$$

where we recognize that the boundary condition on v_1 requires

$$\hat{v} = 0 \text{ at } y = \pm L, \quad (16)$$

which expresses that even in inviscid flows, fluid can't go into or come out of boundaries.

Rayleigh was able to take the problem defined by (15) and (16) to get a condition for the stability of this system without actually solving the equation (15). First, note that since \hat{v} is complex, we can write

$$\hat{v} = \hat{v}_R + i\hat{v}_I,$$

and we write the *complex conjugate* of \hat{v} as

$$\tilde{v} = \hat{v}_R - i\hat{v}_I.$$

Then, \hat{v} and its complex conjugate \tilde{v} satisfy

$$\tilde{v}\hat{v} = |\hat{v}|^2.$$

What Rayleigh did is to multiply (15) by \tilde{v} , then integrate across the domain in the y -direction:

$$\int_{-L}^L \tilde{v}\hat{v}'' dy + \int_{-L}^L \left(\frac{kU''}{\omega - Uk} - k^2 \right) |\hat{v}|^2 dy = 0. \quad (17)$$

We attack the first integral by integrating by parts:

$$\int_{-L}^L \tilde{v}\hat{v}'' dy = \overbrace{[\hat{v}'\tilde{v}]_{-L}^L}^{\text{use (16)}} - \int_{-L}^L |\hat{v}'|^2 dy = - \int_{-L}^L |\hat{v}'|^2 dy$$

so (17) becomes

$$-\int_{-L}^L |\hat{v}'|^2 dy + \int_{-L}^L \left(\frac{kU''}{\omega - Uk} - k^2 \right) |\hat{v}|^2 dy = 0. \quad (18)$$

Now, remember that since ω is complex, we can write

$$\omega = \omega_R + i\omega_I$$

and using the complex conjugate and noting that U and k are real, we also have

$$(\omega - Uk)(\tilde{\omega} - Uk) = |\omega - Uk|^2,$$

which allows us to write

$$(\omega - Uk) = \frac{|\omega - Uk|^2}{\omega_R - i\omega_I - Uk}. \quad (19)$$

Then, plugging (19) into (18), we have

$$-\int_{-L}^L |\hat{v}'|^2 dy + \int_{-L}^L \left[\frac{(\omega_R - i\omega_I - Uk)kU''}{|\omega - Uk|^2} - k^2 \right] |\hat{v}|^2 dy = 0. \quad (20)$$

We can see why we wrote (19) by noting that in (20), the real and imaginary parts can easily be separated, which wouldn't be the case if there were complex terms in the denominator of the fractional term.

In (20), the real and imaginary parts of the left side have to be zero individually. Look at the imaginary part:

$$\omega_I k \int_{-L}^L \frac{U'' |\hat{v}|^2}{|\omega - Uk|^2} dy = 0. \quad (21)$$

Recall from our discussion of surface waves that disturbances like (11), where k is real and ω complex, grow exponentially in time if $\omega_I > 0$. If this is true, then the integral in (21) has to be zero. But everything in the integrand is necessarily positive everywhere in the domain of integration except for U'' . So for the integral to be zero, U'' either has to be zero everywhere, or it has to be positive in part of the domain and negative in another part. That is, if we consider profiles where U'' is continuous, then for $\omega_I > 0$ to hold, the velocity has to have an inflection point (where $U'' = 0$). This is Rayleigh's theorem:

- **Rayleigh's Inflection Point Theorem.** A necessary condition for the linear instability of an inviscid shear flow $U(y)$ is that $U''(y)$ must change sign somewhere in the flow, i.e. the velocity profile $U(y)$ must have an inflection point.

This is only a *necessary* condition for linear instability, so just because a profile has an inflection point doesn't mean that particular parallel inviscid flow is unstable. But we *can* use Rayleigh's theorem to get a necessary condition for stability (meaning that disturbances don't grow in time); in particular, if a flow doesn't have an inflection point, then it is stable.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 33 - Instability: parallel shear flow, transition to turbulence

Reading: Acheson, §9.5, 9.8.

In the last lecture we investigated the stability of parallel shear flows of ideal fluids, and proved Rayleigh's theorem, which says that a necessary condition for the linear instability of an inviscid shear flow $U(y)$ is that the $U(y)$ profile must have an inflection point. What happens when we consider viscous fluids?

As before, suppose we have two plane boundaries at $y = -L$ and $y = L$, and we consider two-dimensional flows, $\mathbf{u} = (u, v, 0)$, for which the Navier-Stokes equations take the following form:

$$\begin{aligned}\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} &= -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \\ \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} &= -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \\ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0,\end{aligned}\tag{1}$$

and because the fluid is viscous, we also have the no-slip condition

$$u = 0 \text{ at } y = \pm L.\tag{2}$$

As in the inviscid case, we will consider an undisturbed flow given by

$$\mathbf{u}_0 = (U(y), 0, 0),$$

and we will look at small perturbations to this flow given by

$$\mathbf{u}_1 = (u_1(x, y, t), v_1(x, y, t), 0)$$

We'll do one thing differently from the viscous case; that is, we'll express the disturbances \mathbf{u}_1 in terms of a stream function ψ , specifically

$$u_1 = \frac{\partial \psi}{\partial y} \text{ and } v_1 = -\frac{\partial \psi}{\partial x}$$

and we look at

$$\psi = \text{Re} \left[\hat{\psi}(y) e^{i(kx - \omega t)} \right]$$

where k is real and $\hat{\psi}$ and ω are complex. As before when we consider this type of periodic disturbance, the imaginary part of $\omega = \omega_R + i\omega_I$ determines the (temporal) stability of the problem. If $\omega_I > 0$ then the disturbance grows exponentially in time (i.e. the flow is unstable to the disturbance), if $\omega_I < 0$ the disturbance decays exponentially in time (the flow is stable to the disturbance), and if $\omega_I = 0$, the disturbance remains constant in magnitude (this condition is known as *marginal stability*).

The boundary conditions on the disturbances are that the v -component at the boundaries needs to be zero, as before, and also that the no-slip condition needs to be observed at the boundaries, since the fluid is viscous. Thus

$$\begin{aligned}v_1 = 0 &\rightarrow \hat{\psi} = 0 \text{ at } y = \pm L, \text{ and} \\ u_1 = 0 &\rightarrow \hat{\psi}' = 0 \text{ at } y = \pm L.\end{aligned}$$

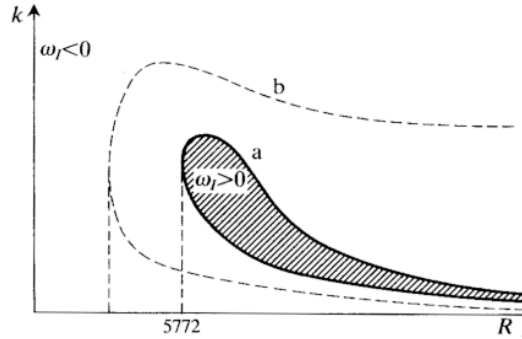


Figure 1: Stability curves for plane Poiseuille flow (a), from Acheson (Fig. 9.11). See the text for an explanation of curve (b).

Inserting these forms for the velocity and pressure disturbances in the Navier-Stokes equations (1), we eventually get the *Orr-Sommerfeld* equation –

$$i\nu(\hat{\psi}'''' - 2k^2\hat{\psi}'' + k^4\hat{\psi}) + (Uk - \omega)(\hat{\psi}'' - k^2\hat{\psi}) - U''k\hat{\psi} = 0. \quad (3)$$

Acheson discusses the particular case of plane Poiseuille flow, in which the undisturbed flow between the boundaries is driven by a constant pressure gradient. In that case, the undisturbed velocity profile, $U(y)$ is:

$$U(y) = U_{\max} \left(1 - \frac{y^2}{L^2} \right). \quad (4)$$

Acheson then presents the stability results for this case (the math itself is, again, beyond us) as a plot (given as Fig. 1) of wavenumber, k , vs. the Reynolds number, which is defined for this problem as

$$Re = U_{\max}L/\nu. \quad (5)$$

The key observation from the plot is that there *is* a region of wavenumbers and Reynolds numbers for which the flow is unstable. In the inviscid case, the profile (4) is stable, because it has no inflection point (the second derivative is never zero). So in the viscous case, the existence of viscosity can actually *destabilize* the flow! This sort of runs counter to our intuition, which is that the the ability of a viscous flow to dissipate kinetic energy through friction ought to help damp out disturbances, and thus stabilize the flow.

There are other observations we can make from Fig. 1. For one, we can see one sense in which viscosity does have a stabilizing effect. There is a critical value of the Reynolds number, which Acheson gives as $Re_c = U_cL/\nu = 5772$ for plane Poiseuille flow, below which the flow is stable to disturbances of all wavenumbers. Since the Reynolds number is defined by (5), increasing the viscosity, ν , stabilizes the flow in the sense that the critical velocity, U_c , is increased. Also, we can see from the figure that as we increase the Reynolds number, i.e. as viscosity becomes less important relative to inertial forces, the range of wavenumbers k for which the flow is unstable becomes smaller, eventually disappearing as $Re \rightarrow \infty$. Thus, the inviscid stability solution from our previous lecture can be treated as the limiting case of the viscous stability solution for $\nu \rightarrow 0$.

1 Turbulence

The ideas of instability are understood to explain the onset of turbulence, at least in a certain sense. In the context of Reynolds's experiment of flow in a tube, we know that the flow is subject

to background disturbances. For low enough Reynolds numbers, these disturbances are unable to affect the flow structure, but as the Reynolds number increases, the disturbances grow and cause the flow structure to make the transition to turbulence. Of course, there are complicating factors. For starters, we have dealt only with infinitesimal disturbances, for which we can apply linear approximations; specifically, we have been making the assumption that products of perturbation terms are negligible relative to terms that are linear in perturbations. We then tracked the time evolution of these disturbances when applied to a particular background flow (the same principles could have been applied to tracking the spatial evolution of the disturbances). In the event that the amplitude of the disturbance grew with time, we said that the background flow was *unstable* relative to that disturbance, while in the event that the disturbance either decayed or remained of constant amplitude with time, we said that the flow was *stable* relative to the disturbance.

We can see immediately that there are limitations to this view of things. For example, it is invalid to apply linear theory once the disturbance magnitudes are not infinitesimal, so technically, we can't say with certainty that a disturbance that initially grows in time will continue to do so for all time. As a disturbance grows, we start to have to consider nonlinear effects; and *a priori*, we have no way to predict if the disturbance will continue to grow with time when nonlinearity is considered. There has been a great deal of work in nonlinear instability theory (see, for example, the book by Drazin and Reid that's on reserve). However, the particular link between flow instability and turbulence is still not well understood.

Actually there have been a multitude of attempts to 'explain' turbulence theoretically, and still we search for a satisfactory 'solution.' 'Solution' appears in quotation marks because it is not, in fact, easy to say exactly what it would mean to 'solve' turbulence. Consider the following analysis.

Recognizing that turbulence is in some sense a random process superimposed on some orderly flow arrangement, it is customary in discussing turbulence to separate flow quantities in turbulence into their mean and fluctuating parts. For the velocity field, this means

$$\mathbf{u}(\mathbf{x}, t) = \langle \mathbf{u}(\mathbf{x}, t) \rangle + \mathbf{u}'(\mathbf{x}, t), \quad (6)$$

where $\langle \cdot \rangle$ denotes some averaging process (which, depending on the flow of interest, could be in time or space, for now it doesn't matter which) and the primed term is the fluctuating term. Clearly, the fluctuating term describes the deviation of the actual value of the velocity vector from the mean value, at a particular point in space and time (\mathbf{x}, t) . The form (6) is known as the *Reynolds decomposition*. We point out for now some properties of the mean. The mean of a sum is equal to the sum of the means, or

$$\langle a + b \rangle = \langle a \rangle + \langle b \rangle,$$

(where a and b are functions that can vary in space and time); taking the mean of a mean leaves it unchanged, or

$$\langle \langle a \rangle \rangle = \langle a \rangle,$$

and the mean commutes with both temporal and spatial differentiation, so

$$\begin{aligned} \left\langle \frac{\partial a}{\partial t} \right\rangle &= \frac{\partial \langle a \rangle}{\partial t} \text{ and} \\ \left\langle \frac{\partial a}{\partial x_j} \right\rangle &= \frac{\partial \langle a \rangle}{\partial x_j}. \end{aligned} \quad (7)$$

What happens to the Navier-Stokes equations when we write the velocity in the decomposed form (6)? For the continuity condition, $\nabla \cdot \mathbf{u} = 0$, we get, taking the average—

$$\langle \nabla \cdot \mathbf{u} \rangle = \nabla \cdot \langle \mathbf{u} \rangle = 0 \quad (8)$$

since averaging commutes with differentiation. We can also show easily from this that

$$\nabla \cdot \mathbf{u}' = 0. \quad (9)$$

So (8) and (9) tell us that the mean and fluctuating parts of the velocity field each satisfy the continuity condition.

More interesting is what happens to the equation for the momentum, which is in its original form

$$\frac{Du_i}{Dt} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \nabla^2 \mathbf{u}_i. \quad (10)$$

Now write the material derivative as

$$\frac{Du_i}{Dt} = \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = \frac{\partial u_i}{\partial t} + \frac{\partial(u_i u_j)}{\partial x_j}$$

where the second step uses the product rule and the continuity condition. Now take the mean of this:

$$\left\langle \frac{Du_i}{Dt} \right\rangle = \frac{\partial \langle u_i \rangle}{\partial t} + \frac{\partial \langle u_i u_j \rangle}{\partial x_j},$$

and substitute (6), which leads us to

$$\left\langle \frac{Du_i}{Dt} \right\rangle = \frac{\partial \langle u_i \rangle}{\partial t} + \langle u_j \rangle \frac{\partial \langle u_i \rangle}{\partial x_j} + \frac{\partial \langle u'_i u'_j \rangle}{\partial x_j}. \quad (11)$$

Finally, taking the mean of (10) and substituting (11), we get

$$\frac{\partial \langle u_i \rangle}{\partial t} + \langle u_j \rangle \frac{\partial \langle u_i \rangle}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \langle p \rangle}{\partial x_i} + \nu \nabla^2 \langle u_i \rangle - \frac{\partial \langle u'_i u'_j \rangle}{\partial x_j}. \quad (12)$$

Now suppose you're someone whose job is to compute turbulence numerically. As we will see, even when the mean terms vary slowly in space (and time), the fluctuating terms can vary over very small length and time scales. In fact, in general, it is unrealistic (in terms of computational resources) to compute all of the length scales in turbulence directly, because the required computational grid would have an unmanageable number of points. The problem, relative to the decomposed Navier-Stokes equation (12), becomes, how can we relate the term

$$\frac{\partial \langle u'_i u'_j \rangle}{\partial x_j},$$

which is known as the *Reynolds stress*, and which varies on the smallest length and time scales of the flow, to quantities like $\langle u_i \rangle$ that we can compute on reasonably large scales? This problem (or some closely related form of it) has occupied a lot of people for a long time now.

Ultimately, this problem (and, in fact, most of the difficulty with studying turbulent flows) arises from the wide disparity in the length scales of interest in turbulence. To get some feel for this aspect of turbulent flows, consider Figs. 2 and 3.

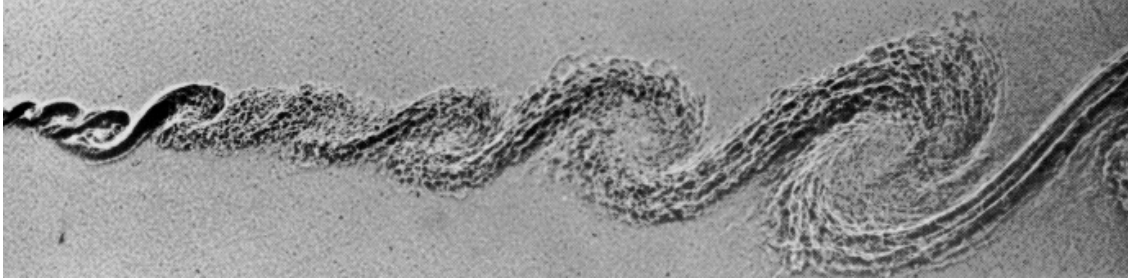


Figure 2: A mixing layer at high Reynolds number. The upper stream is moving at 100 m/s and the lower at 38 m/s, both from left to right in the image. Two things: (1) the transition to turbulence is evident on the left side of the image, where the initially smooth roller structures suddenly develop small-scale detail, and (2) the large-scale organization of the flow is evident as the flow moves downstream, even though the flow has a lot of small-scale activity.

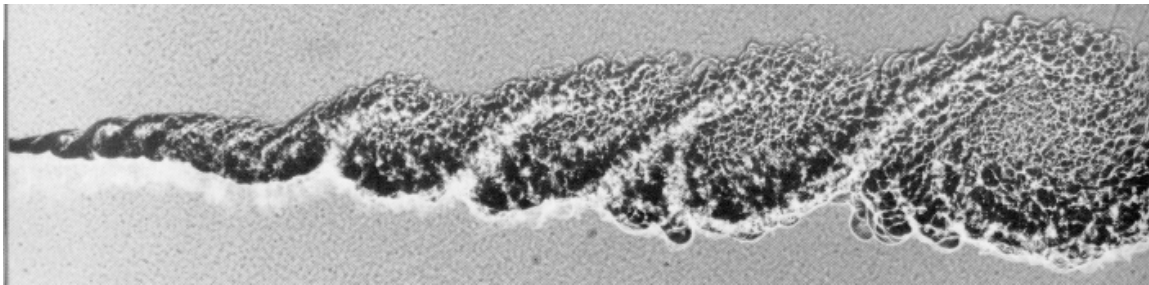


Figure 3: The same flow arrangement as above, but at double the Reynolds number. There is more small-scale activity, but the large-scale organization is not greatly affected.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 34 - Turbulence: basic ideas

We're not going to have time to discuss turbulence in any level of detail, so we'll just touch on a few significant ideas. When we speak of turbulence, certain things should come to mind:

- 'Randomness' and disorder at small flow scales
- Coherence and order at larger flow scales
- Energy cascade: transfer of energy between length scales.
- Similarity and scaling of flow properties.

We'll talk about these things briefly in the time remaining.

1 Randomness and disorder - statistical tools

When we speak of 'randomness,' we have a particular definition in mind. Saying for example that the velocity, \mathbf{u} , is random doesn't mean that it can take on any value; instead, it means that any time we run an experiment under the same conditions, \mathbf{u} will take on a different value.

It may seem strange that \mathbf{u} can be a random variable when the time evolution of \mathbf{u} is described by a deterministic system of equations, the Navier-Stokes equations. The Navier-Stokes equations have the form

$$\frac{\partial \mathbf{u}}{\partial t} = (\text{something}),$$

and, in principle, we should be able to integrate this equation in time, subject to initial and boundary conditions, to get the solution, $\mathbf{u}(\mathbf{x}, t)$, for the velocity at all later times. So where does the randomness come from? Well, as we discussed in the context of stability theory, there will always be perturbations in the initial and boundary conditions of the flow – walls are not perfectly still or flat, pressure is never exactly uniform, etc. Then on top of that, turbulent flows are particularly sensitive to perturbations (think of the exponential growth of disturbances in unstable flows).

To describe the randomness of variables like the velocity component u , we use something called the *probability density function*, or *PDF*, which is itself definable in terms of the cumulative distribution function, or *CDF*. We start this discussion (which follows the notation of the book *Turbulent Flows* by S. B. Pope) with the following definition. If we run an experiment and monitor the velocity at a particular point, the probability, P , that u will be less than the value V is

$$F(V) = P(u < V) \tag{1}$$

(V is a dummy variable called the sample-space variable, *not* the y -component of velocity). The function F is the CDF. There are three basic properties to the CDF:

$$\begin{aligned} F(-\infty) &= 0, \\ F(\infty) &= 1, \\ V_b > V_a &\Rightarrow F(V_b) \geq F(V_a). \end{aligned} \tag{2}$$

The first two of these tell us that u can't be less than $-\infty$, and that u has to be less than ∞ , respectively. To understand the last property, observe that, based on the definition (1), the

probability that u is between V_a and $V_b > V_a$ is given by the probability that $u < V_b$ minus the probability that $u < V_a$, or

$$P(V_a < u < V_b) = F(V_b) - F(V_a).$$

Since the probability of an event can't be less than zero, this last expression means that $F(V_b) \geq F(V_a)$.

Now we define the probability density function (PDF), as the derivative of the CDF –

$$f(V) = \frac{dF(V)}{dV}. \quad (3)$$

From the properties (2) of the CDF, we can derive the following properties for the PDF. First,

$$f(V) \approx \frac{F(V + dV) - F(v)}{dV} \geq 0$$

using the third property in (2). Second,

$$\int_{-\infty}^{\infty} f(V)dV = f(V = \infty) - f(V = -\infty) = 1.$$

Finally,

$$P(V_a < u < V_b) = F(V_b) - F(V_a) = \int_{V_a}^{V_b} f(V)dV. \quad (4)$$

If we let V_b and V_a be separated by the infinitesimal interval dV in (4), we get

$$P(V < u < V + dV) = F(V + dV) - F(V) = \frac{dF}{dV}dV = f(V)dV, \quad (5)$$

which expresses that $f(V)$ is the probability per unit (whatever the random variable is, in this case velocity), which is why we call it the probability *density* function.

The form of the PDF, f , completely characterizes the random variable u . In practice we are often interested in quantities like the mean. The mean of the random variable u is written

$$\langle u \rangle = \int_{-\infty}^{\infty} V f(V)dV \quad (6)$$

i.e. it is the probability-weighted average of all possible u values. In general, if $G(u)$ is some function of u , we can write the mean of G as

$$\langle G(u) \rangle = \int_{-\infty}^{\infty} G(V)f(V)dV. \quad (7)$$

It's worth thinking a little bit about how to interpret the mean defined by (6). $\langle u \rangle$ can be thought of as the *expected* value of u . For example, if we run our experiment N times with the same conditions, and measure u at the same spatial (and temporal) location, then as N gets very large, the average of our observations of u will approach $\langle u \rangle$. (So it's not really a spatial average or a time average, although for certain flow conditions, it could be either).

The form of (7) makes it easy to see why the mean of a sum of random variables is equal to the sum of the means. Suppose G and H are both functions of the random variable u (which makes G and H also random variables). Then the mean of $G + H$ is

$$\langle G(u) + H(u) \rangle = \int_{-\infty}^{\infty} (G(V) + H(V))f(V)dV = \int_{-\infty}^{\infty} G(V)f(V)dV + \int_{-\infty}^{\infty} H(V)f(V)dV = \langle G \rangle + \langle H \rangle.$$

Also, it is easy to see that the mean of a random variable isn't a random variable. Again, look at (7). Clearly, $\langle G(u) \rangle$ is not a function of u (and thus not a random variable), because the u -dependence has been integrated out. So we can write

$$\langle \langle G(u) \rangle \rangle = \int_{-\infty}^{\infty} \langle G \rangle f(V)dV = \langle G \rangle \int_{-\infty}^{\infty} f(V)dV = \langle G \rangle.$$

In the last lecture we touched on the idea of the decomposing u using the mean:

$$u = \langle u \rangle + u'. \quad (8)$$

One of the interesting things to look at in turbulence is the magnitude of u' , which tells us in some sense just how random turbulence is. To quantify the magnitude of u' , we might try just to look at its mean value. But if we take the mean of (8), we get

$$\langle u \rangle = \langle u \rangle + \langle u' \rangle$$

(using $\langle \langle u \rangle \rangle = \langle u \rangle$), which gives us

$$\langle u' \rangle = 0$$

i.e. the mean value of the fluctuating term in (8) is zero, which should be no surprise. To measure the magnitude of u' , we instead use the *variance*, which is

$$\langle u'^2 \rangle = \int_{-\infty}^{\infty} (V - \langle u \rangle)^2 f(V)dV.$$

The square root of this is called the standard deviation, sometimes seen as σ_u , and often called the 'r.m.s.' (root-mean-square, referring to how it's computed) of u . We can quantify the intensity of the turbulence by looking at the value of σ_u . In a pipe flow, for example, we might compare σ_u to the mean velocity in the pipe, $\langle u \rangle$. If σ_u is on the order of 10% of the mean flow velocity, that's pretty turbulent.

Finally, using the definition of the mean, we can show easily that the mean commutes with differentiation. For time differentiation, first write

$$\frac{du(t)}{dt} = \lim_{dt \rightarrow 0} \frac{u(t + dt) - u(t)}{dt}$$

then write

$$\begin{aligned} \left\langle \frac{du(t)}{dt} \right\rangle &= \left\langle \lim_{dt \rightarrow 0} \frac{u(t + dt) - u(t)}{dt} \right\rangle \\ &= \lim_{dt \rightarrow 0} \frac{1}{dt} \langle u(t + dt) - u(t) \rangle \\ &= \lim_{dt \rightarrow 0} \frac{\langle u(t + dt) \rangle - \langle u(t) \rangle}{dt} \\ &= \frac{d\langle u(t) \rangle}{dt}. \end{aligned} \quad (9)$$

The same argument pertains to spatial differentiation.

It is important to keep in mind that in turbulence, as we saw in the figures in Lecture 33, the randomness is most evident at small scales. Therefore the fluctuation terms like u' tend to vary on small flow scales, while mean terms like $\langle u \rangle$ tend to vary over much larger scales. This insight motivates virtually all theoretical approaches to understanding turbulence.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 35 - Turbulence: the energy cascade, scaling concepts

Now that we've seen that turbulent flows tend to be characterized simultaneously by large-scale organization and a high degree of small-scale disorder, we might reasonably ask what sort of connection there might be between these scales. This question was first addressed by Richardson in the 1930s, and was subsequently taken up also by the mathematician Kolmogorov in the following decade. In this discussion we will be interested in very high Reynolds number turbulence, also called fully-developed turbulence, for which the notion of a wide disparity between the large and small scales is well justified.

1 The energy cascade

Richardson conceived of turbulence as being composed of 'eddies' of different sizes. What is an eddy? Though it isn't strictly correct, we can imagine eddies as being individual vortices of some measurable diameter, l . (As an alternative, Pope offers the following in his book *Turbulent Flows*; "An 'eddy' eludes precise definition, but it is conceived to be a turbulent motion, localized within a region of size l , that is at least moderately coherent over this region.") Richardson then had the idea that large eddies are prone to break into smaller eddies, which break up into smaller eddies, etc. etc. In each breakup step, the larger eddy transfers its energy to the smaller ones, without dissipation, meaning that the energy transfer process in turbulence is *inviscid*. This makes perfect sense, since turbulence requires high Reynolds numbers, which in turn mean that viscous forces are negligible (with respect to inertial forces).

To complete this picture, we need to say what happens at the ends of the process. Energy can't just be transferred from large scales to smaller scales forever. Somehow energy has to be put into the system, and somehow, energy has to be taken out. We know that energy dissipation is a viscous process, and we know that the energy transfer process is inviscid, so this tells us that energy is only taken out of the system at the smallest length scales, and that viscosity is important only at those scales. Meanwhile, the process is kicked off by the break-up of the largest eddies. These large eddies have the same characteristic velocity scale, U , and length scale, which we'll call δ , as the overall flow does. Using only U and δ , we can form a rate of energy transfer as U^3/δ (to see why, verify the units). This represents the rate at which energy is put into the transfer process by the breakup of the large eddies.

Figure 1 outlines this energy cascade process. We denote the rate at which energy is put into

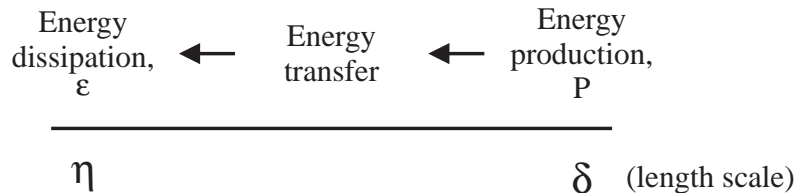


Figure 1: The energy cascade process in fully-developed turbulence. The length scale η represents a characteristic scale of energy-dissipating eddies – we'll discuss this later.

the system by P , and the rate at which energy is dissipated as ε . The energy input P determines ε . They are *not*, however, generally equal, though you might think they need to be by an energy balance. Keep in mind that the energy we're talking about is the energy contained in the turbulent motions only. There is, in addition, energy in the flow that is contained in the *mean* part of the

flow. The production term P really represents energy that is taken in some sense from the mean flow.

This last point gives us insight into exactly *how* energy enters the system. The scales U and δ represent a level of mean shear, if we view δ as a distance over which the mean velocity can vary by U . Since U and δ are sufficient to define the rate of energy input to the system, we can say that energy is put into the system by the mean shear of the flow.

There is an interesting analogy to the turbulent transfer process that exploits this observation. Consider a system of two parallel plates, between which sit a bunch of rocks with characteristic size roughly equal to the distance between the plates. Now suppose we allow the plates to move tangentially and in opposite directions to each other, generating a shear on the rocks. Eventually the big rocks break to form smaller ones, until finally you're left with dust that can't be made any smaller (Fig. 2). (Descriptions of this process are called breakage models.) Considering that energy

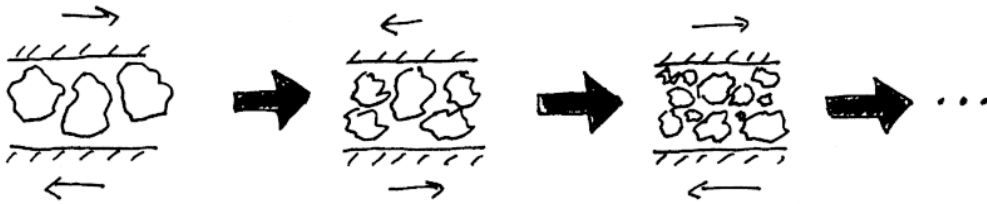


Figure 2: The rock breakup process.

is put into the system through the shearing of the rocks, one can ask questions like, what is the distribution of rock sizes at any point in time?

2 Kolmogorov's hypotheses

Kolmogorov came to the subject of turbulence partly because of an interest in associated problems like the rock breakage problems. In 1941, he wrote a paper, barely more than two pages long, in a Russian mathematics journal that has gone on to spawn an amazing amount of work. That original paper and its follow-ups addressed questions like:

- What is the size of the smallest length scales in turbulence, i.e. the ones that are responsible for dissipating energy?
- What's the relation between the size, l , of an eddy and the characteristic velocity, $u(l)$, associated with it?

In attacking these problems, Kolmogorov came up with three hypotheses.

- **Kolmogorov's hypothesis of local isotropy.** For high enough Reynolds number $Re = U\delta/\nu$, the small-scale turbulent eddies (with $l \ll \delta$) are statistically isotropic.

What this means is, while the large scales of the flow are permitted to look different in different directions (i.e., to be anisotropic), with properties defined by the mean flow field and the boundary conditions, the small scales will look the same in all directions. ('Local isotropy' means that the isotropy is confined to small scales.) Kolmogorov argued that, in proceeding down the energy cascade, all of the directional information in the large-scale motions is lost.

- **Kolmogorov’s first similarity hypothesis.** For high enough Re , the statistics of the small-scale eddies ($l \ll \delta$) take on a universal form determined by ν and ε , the energy dissipation rate. We can label these length scales as $l \approx \eta$ (where we will define η later).

This is basically a stronger version of the isotropy hypothesis. Just as the directional information in the large scales is lost in the energy cascade process, so will all information about the geometry of the large scales, which are set by the mean flow and the boundary conditions. We are left with small scales that don’t know anything about the large scales. This means, in other words, that the small scales are somehow universal and should be similar in *every* turbulent flow with high enough Reynolds number. Since energy is dissipated at these small scales, the important parameters are (1) the rate at which energy comes to the small scales, which (it turns out) is given by ε , and (2) the viscosity ν , since energy dissipation is a viscous process.

- **Kolmogorov’s second similarity hypothesis.** For high enough Re , the statistics of motion in the range between δ and the dissipation scale η (specifically, for length scales l such that $\eta \ll l \ll \delta$) have a universal form that is determined by ε and is independent of ν .

This hypothesis comes from the idea that if the Reynolds number is high enough, so the length scales δ and η are really widely separated, there will be eddies of a size sufficiently smaller than δ that they will retain no memory of the large scale properties of the flow, yet they will be sufficiently larger than η that the viscosity will still be unimportant.

3 Aside: shear flow scaling

We’ll take a moment to discuss an interesting way that we can use some of these ideas, along with some ideas we’ve seen earlier in the class. Let’s look at an axisymmetric turbulent jet (Fig. 3). In this flow, there is a point source of momentum with a momentum flux J_0 , which has units of

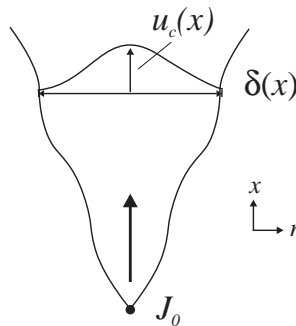


Figure 3: An axisymmetric turbulent jet.

momentum per unit time, in the x -direction. The jet fluid and ambient fluid both have density, ρ_0 , and viscosity, ν_0 .

If the momentum flux is high enough, the jet will be turbulent. Suppose now that we’re only interested in the large-scale, mean flow properties. In an earlier homework assignment we worked the problem of a laminar jet using the boundary layer equations. A key aspect of that solution was that the velocity profile was self-similar. We can apply the same principle here, and by the geometry of the problem we will also assume the the flow is axisymmetric. Specifically, we have

$$u(x, r) = u_c(x)f(\eta), \text{ where } \eta = \frac{r}{\delta(x)},$$

where $u(x, r)$ is the *mean* axial velocity. (We're using cylindrical coordinates, but with x instead of z as the axial coordinate.) What we want to know is, how do u_c and δ depend on x ?

Finding the flow width, δ , is easy. The major assumption is taken from the discussion of the previous section. In turbulent flows, the importance of viscosity is basically confined to the smallest scales, and since we're interested in the large-scale properties, this means we can ignore viscosity. So we can say that δ depends on x , J_0 and ρ_0 . Considering the units of these:

$$\delta \rightarrow \text{length}, x \rightarrow \text{length}, J_0 \rightarrow \frac{\text{mass} \cdot \text{length}}{\text{time}^2}, \rho_0 \rightarrow \frac{\text{mass}}{\text{length}^3}.$$

The units of δ don't involve time, so δ can't depend on J_0 ; but then δ also doesn't involve mass, so we can't use ρ_0 either. Thus we just end up with

$$\delta = Cx.$$

To find the x -dependence of u_c , we make use of the conservation of momentum, since there aren't any forces in our problem (why can't we use the conservation of mass?). The momentum flux through any arbitrary $x = \text{const.}$ surface has to equal J_0 , or

$$\begin{aligned} J_0 &= J(x)\rho_0 \int_0^{2\pi} d\theta \int_0^\infty dr u^2 r \\ &= 2\pi\rho u_c^2 \int_0^\infty \delta d\eta f^2 \delta \eta \\ &= 2\pi\rho u_c^2 \delta^2 \int_0^\infty \eta f^2 d\eta. \end{aligned}$$

Looking at the last equality, the integral is constant, since f is a function only of η . So $u_c^2 \delta^2$ has to be a constant, which means that

$$u_c = \frac{D}{x}.$$

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 36 - Turbulence: more on scaling, and the Reynolds stress

In the last class we looked at the Kolmogorov similarity hypotheses, which expressed the idea that the smaller scales of turbulence approach a universal state for high enough Reynolds number. (The Reynolds number we're interested in is $Re = u_0 l_0 / \nu$, where u_0 is a characteristic velocity of the flow large scale, and l_0 is a characteristic large-scale flow dimension.) That is, while the large scales will obviously be different, being set by the different flow boundary conditions, etc., the statistics of the small scales of the flow will be the same between flows if the Reynolds numbers are high enough. Also, if you look at the smallest turbulent scales, the turbulence at those scales will be isotropic, meaning that all of the spatial directions will look the same.

Based on Kolmogorov's hypotheses, we can also estimate the size of the smallest turbulence length, time and velocity scales. From Kolmogorov's first similarity hypothesis, the universal form of the *smallest* turbulence scales is dependent only on the viscosity, ν and the rate of energy dissipation, ε . The units of these terms are

$$\nu = \left[\frac{\text{length}^2}{\text{time}} \right], \quad \varepsilon = \left[\frac{\text{velocity}^2}{\text{time}} \right] = \left[\frac{\text{length}^2}{\text{time}^3} \right] \quad (1)$$

where we'll note that ε is really the *specific* rate of energy dissipation, i.e. the rate per unit mass. From these, we can form a unique length scale, η , time scale, τ_η , and velocity scale, u_η , as:

$$\begin{aligned} \eta &= \left(\frac{\nu^3}{\varepsilon} \right)^{1/4} \\ \tau &= \left(\frac{\nu}{\varepsilon} \right)^{1/2} \\ u_\eta &= (\varepsilon \nu)^{1/4}. \end{aligned} \quad (2)$$

These are collectively called the *Kolmogorov scales*, though if you see reference to 'the Kolmogorov scale,' usually this means just the length scale, η .

From the scales (2), we can form the Reynolds number

$$Re_\eta = \frac{u_\eta \eta}{\nu} = 1, \quad (3)$$

which tells us that at the Kolmogorov scales, inertial effects and viscous effects are of roughly equal importance. This is consistent with the idea that the the scales (2) represent the smallest scales of turbulence, and with the notion of the energy cascade we also discussed earlier. Remember that we said that in a turbulent flow, energy is transferred without dissipation to successively smaller scales until, at the smallest scales, viscosity becomes important and energy is dissipated.

The scales defined by (2) give us useful insight, but in reality, the dissipation ε is very difficult to measure experimentally. It would be more convenient if we could express the Kolmogorov scales in terms of large-scale quantities. In particular, we'd like to express η , τ_η and u_η in terms of the large-scale length and velocity scales l_0 and u_0 . To do this, we remember from our prior discussion (Lecture 34) that the dissipation rate scales with l_0 and u_0 as

$$\varepsilon \propto \frac{u_0^3}{l_0}, \quad (4)$$

which told us that the energy dissipated by a turbulent flow is proportional to the energy that's

put into the cascade process at the large scales. Using (4) in (2), we get the relations

$$\begin{aligned}\frac{\eta}{l_0} &\propto Re^{-3/4} \\ \frac{u\eta}{u_0} &\propto Re^{-1/4} \\ \frac{\tau_\eta}{(l_0/u_0)} &\propto Re^{-1/2}\end{aligned}\tag{5}$$

where $Re = u_0 l_0 / \nu$. From (5), it is clear that as the Reynolds number increases, the disparity between the large and small scales increases. For example, if we take a given flow system and bump up the Reynolds number by increasing the velocity, the small length scale will get smaller even as the large scale l_0 stays the same.

1 The Reynolds stress

Recall from Lecture 34 that if we decompose the velocity field in turbulent flows as

$$u_i = \langle u_i \rangle + u'_i,\tag{6}$$

and also write

$$p = \langle p \rangle + p',\tag{7}$$

then the evolution equation for $\langle u_i \rangle$ becomes

$$\frac{\partial \langle u_i \rangle}{\partial t} + \langle u_j \rangle \frac{\partial \langle u_i \rangle}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \langle p \rangle}{\partial x_i} + \nu \nabla^2 \langle u_i \rangle - \frac{\partial \langle u'_i u'_j \rangle}{\partial x_j},\tag{8}$$

which we compare to the Navier-Stokes equation for the raw u_i :

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \nabla^2 u_i.\tag{9}$$

The difference is obviously the last term in (8), which we'll write as

$$\frac{\partial R_{ij}}{\partial x_j}, \text{ where } R_{ij} = \langle u'_i u'_j \rangle.\tag{10}$$

(Note: calling this term R_{ij} is not standard notation.)

It goes without saying that this term $\partial R_{ij} / \partial x_j$ is significant. From (8) and (9), the raw field u_i and the mean field $\langle u_i \rangle$ would be identical except for $\partial R_{ij} / \partial x_j$. This means that all of the disorder that a turbulent flow manifests can be traced to $\partial R_{ij} / \partial x_j$. This is a pretty strong statement.

(Consider Fig. 1, which shows the scalar concentration of a turbulent water jet. (The scalar in this case is a laser fluorescent dye in the jet fluid.) The mean scalar field is nice and smooth, basically Gaussian in profile. All of the disorder that you see is due to the scalar equivalent of the term $\partial R_{ij} / \partial x_j$.)

What exactly does this $\partial R_{ij} / \partial x_j$ represent? From our discussion of the Navier-Stokes equations, remember that (ignoring gravity)

$$\frac{Du_i}{Dt} = \frac{1}{\rho} \frac{\partial T_{ij}}{\partial x_j},\tag{11}$$



Figure 1: The scalar concentration of a turbulent water jet.

where T_{ij} was the stress tensor, given by

$$T_{ij} = -p\delta_{ij} + \mu \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right). \quad (12)$$

Supposing that we're thinking about a small fluid element, what (11) and (12) tell us is that the acceleration of the fluid element is caused by pressure forces acting on the element, and by the transfer of momentum between the element and the surrounding fluid. Now let's look at (8). With (11) and (12) in mind, rewrite (8) as

$$\frac{D\langle u_i \rangle}{Dt} = \frac{1}{\rho} \frac{\partial \tilde{T}_{ij}}{\partial x_j}, \quad (13)$$

where

$$\tilde{T}_{ij} = -\langle p \rangle \delta_{ij} + \mu \left(\frac{\partial \langle u_j \rangle}{\partial x_i} + \frac{\partial \langle u_i \rangle}{\partial x_j} \right) - \rho \langle u'_i u'_j \rangle, \quad (14)$$

and \tilde{T}_{ij} is, effectively, the stress tensor that's relevant for the mean velocity field, $\langle u_i \rangle$ (why can't we call this $\langle T_{ij} \rangle$?). If we're again considering some fluid element, (13) and (14) describe the mean acceleration of the fluid element; and, looking at the terms in (14), the first term tells us that there's acceleration due to the mean pressure, the second term tells us that there's acceleration due to transfer of mean momentum, and the third term tells us that there's acceleration due to momentum transfer by the fluctuating velocity field.

This last point is very significant, because one of the major features of turbulent flows is that they show increased transport of quantities like momentum, scalar concentration etc. In other words, the disorder and small-scale activity in turbulent flows makes it easier for quantities like momentum and scalar concentration to mix.

ME 563 - Intermediate Fluid Dynamics - Su

Lecture 37 - Turbulence: some thoughts

The following is from *A First Course in Turbulence*, by Tennekes and Lumley:

“Most flows occurring in nature and in engineering applications are turbulent. The boundary layer in the earth’s atmosphere is turbulent (except possibly in very stable conditions); jet streams in the upper troposphere are turbulent; cumulus clouds are in turbulent motion. The water currents below the surface of the oceans are turbulent; the Gulf Stream is a turbulent wall-jet kind of flow. The photosphere of the sun and the photospheres of similar stars are in turbulent motion; interstellar gas clouds (gaseous nebulae) are turbulent; the wake of the earth in the solar wind is presumably a turbulent wake. Boundary layers growing on aircraft wings are turbulent. Most combustion processes involve turbulence and even depend on it; the flow of natural gas and oil in pipelines is turbulent. Chemical engineers use turbulence to mix and homogenize fluid mixtures and to accelerate chemical reaction rates in liquids or gases. The flow of water in rivers and canals is turbulent; the wakes of ships, cars, submarines and aircraft are in turbulent motion. The study of turbulence is clearly an interdisciplinary activity, which has a very wide range of applications. In fluid dynamics laminar flow is the exception, not the rule: one must have small dimensions and high viscosities to encounter laminar flow.”

We have discussed turbulence without offering any actual definition for it, because the turbulence is resistant to precise definition. Instead, we have mentioned the following properties.

- **High Reynolds number.** Turbulence occurs at high Reynolds number. Remember that we said that instability of laminar flow was one way that flows become turbulent; instability of a particular flow requires that the Reynolds number exceed some minimum value. The high Reynolds number also means that at the large scales of the flow, inertial forces dominate over viscous forces.
- **Randomness and disorder.** Turbulent flows exhibit a high degree of randomness and disorder, particularly at small scales. We describe these random fluctuations by decomposing the flow variables into mean and fluctuating parts, e.g. $u_i = \langle u_i \rangle + u'_i$.
- **Disparity in length scales.** A wide range of length scales are relevant in turbulent flows. The spread between the largest and smallest length scales in a flow increases with increasing Reynolds number. This makes computation extremely difficult.
- **Energy cascade.** In a turbulent flow, energy generally gets transferred from the large scales to the small scales in an inviscid fashion, then gets dissipated at the small scales by the action of viscosity. The presence of dissipation in turbulent flows is significant; even though the large flow scales may be essentially inviscid, there is viscous dissipation of kinetic energy going on in the flow, at the small scales.
- **Increased transport of momentum, scalars etc.** The random fluctuations of turbulent flows provide another mechanism by which quantities can get transferred from one portion of a flow to another. In a laminar flow, transport of momentum mostly occurs through viscosity (diffusion), while in a turbulent flow, transport can occur through the random motion embodied by the fluctuating terms. Think of a parallel flow, with all the (mean) velocity vectors pointing in the same direction (say, the x -direction). In the laminar case, momentum only gets transferred between layers by viscous drag. In the turbulent case, however, there can be velocity fluctuations in the y -direction to carry momentum across the mean streamlines.

ME 563 - Intermediate Fluid Dynamics - Su
HW 1 - due 4 February

Problem 1 - Cylindrical coordinates. Up to now we've focused on the Cartesian coordinate system. We will also make extensive use of cylindrical coordinates. In cylindrical coordinates, the x - y plane is represented by the radial coordinate r and the angular coordinate θ , and the z -axis is unchanged, as in Fig. 1. The appendix in Acheson's book goes over the vector operations in cylindrical coordinates (Section A.6.)

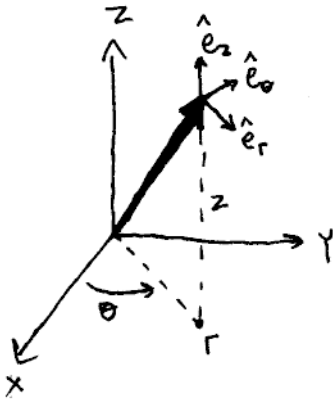


Figure 1: The cylindrical coordinate system.

Here we will familiarize ourselves a little with the cylindrical coordinate system.

- (a) Compute the volume of a right circular cylinder of height h and radius R using cylindrical coordinates. Hint: the coordinate θ does not have dimensions of length.
- (b) The divergence theorem introduced in Lecture 1 is valid for any coordinate system. Using an analysis similar to that in Lecture 1, show that the divergence in cylindrical coordinates is as given in Section A.6 in the text.
- (c) Unlike Cartesian coordinates, where the unit vectors \hat{e}_x , \hat{e}_y and \hat{e}_z have the same orientation at all points in space, the vectors \hat{e}_r and \hat{e}_θ in cylindrical coordinates vary in space. Let \mathbf{s} be the displacement of a point particle relative to the origin. Express \mathbf{s} in cylindrical coordinates. Then, express the velocity of the particle, $d\mathbf{s}/dt$, and its acceleration, $d^2\mathbf{s}/dt^2$, in cylindrical coordinates. Hint: you may want to express \hat{e}_r and \hat{e}_θ in Cartesian coordinates as a convenience.

Problem 2 - Vector identities. (a) Using the list of vector identities in Appendix A in the text, show that when a flow is irrotational, the velocity can be expressed in Cartesian coordinates as

$$\mathbf{u} = \left(\frac{\partial \zeta}{\partial x}, \frac{\partial \zeta}{\partial y}, \frac{\partial \zeta}{\partial z} \right),$$

where ζ is a scalar function known as the *velocity potential*.

(b) Define a *vortex tube* as a thin tube whose surface is parallel to the vorticity vector at every point. Is it possible for a vortex tube to come to an end in a flow, i.e. for the vorticity to go to zero along a surface that cuts through the vortex tube? Why or why not? Is the result limited to ideal fluids?

Problem 3 - Acheson, Exercise 1.1.

Problem 4 - Acheson, Exercise 1.3. Hints: refer to the discussion of Exercise 1.2, and when analyzing the term $(\mathbf{u} \cdot \nabla)\mathbf{u}$, be careful how you handle the unit vectors.

Problem 5 - Acheson, Exercise 1.6.

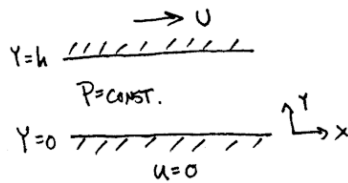
Problem 6 - Acheson, Exercise 1.7, but also show how the 'Lagrangian' description given in the problem is derived.

ME 563 - Intermediate Fluid Dynamics - Su
HW 2 - due 13 February

Problem 1 - Vorticity in viscous fluids.

- (a) Derive the vorticity equation, Eq. (2.39) in the text, starting with Eq. (2.3). Recall that gravity is a conservative force.
- (b) Is the Reynolds number, $Re = UL/\nu$, the proper parameter to describe the relative importance of the inertial and viscous terms in the vorticity equation? Show why or why not.

Problem 2 - Solutions of the Navier-Stokes equations. Consider the steady flow of an incompressible, viscous fluid between two parallel plates, where the the upper plate is moving at speed U in the x -direction, and the lower plate is fixed. The pressure is uniform everywhere in the fluid.



- (a) Solve for the velocity field \mathbf{u} . Clearly state all assumptions.
- (b) Consider the point $x = 0, y = h/2$. What is the shear stress, $\tau_y = \mu \frac{\partial u}{\partial y}$, and what is the magnitude of the viscous term in the Navier-Stokes equation, $\nu \nabla^2 u$, there? Are the results consistent with each other? Explain why or why not.
- (c) Show that the vorticity is nonzero in the fluid. Explain why this is when the flow system is clearly not rotating.

Problem 3 - Acheson, Exercise 2.1. Explain how you estimated the various parameters.

Problem 4 - Acheson, Exercise 2.4. Make sure you understand the relevant example in §2.3 in the text.

Problem 5 - Acheson, Exercise 2.6. Assume the pressure is constant between the plates. You will need to use separation of variables to solve the differential equation - and be careful with your arbitrary constants.

Problem 6 - Acheson, Exercise 2.14. Follow the reasoning in the book's hints carefully. Also, there are four boundary conditions, covering u and v at $y = 0$ and $y \rightarrow \infty$. The condition on v at $y \rightarrow \infty$ is expressed as the similarity function f in terms of η . You will need this to get the expression for the pressure in the hints, which you will need to get the differential equation for f given in the problem.

ME 563 - Intermediate Fluid Dynamics - Su
HW 3 - due 25 February

Problem 1 - Induced velocity fields. Suppose that we have two line vortices, each with circulation Γ and each oriented in the positive z -direction, separated by a distance $2a$ in the $x - y$ plane (see the figure). Assume that the two line vortices are fixed in space. Compute the velocity vectors at the points A through I indicated. Don't work any integrals unless you feel you need to.

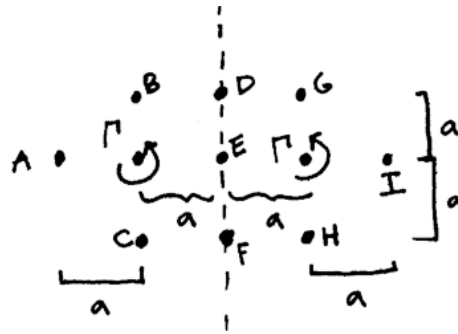


Figure 1: Geometry for Problem 1.

Problem 2. The stream function.

- (a) Write the stream function ψ for the inviscid stagnation point flow $u = \alpha x, v = -\alpha y$. Compute the gradient of the stream function, $\nabla\psi$. Show graphically how the direction of $\nabla\psi$ compares to the streamline directions.
- (b) Write the stream function for the viscous stagnation point flow given in Exercise 2.14.

Problem 3. Acheson, Exercise 4.1. Note: a 'simply connected' region is one whose border is connected, i.e. a simply connected region has no holes.

Problem 4. Acheson, Exercise 5.1. For the circulation computation, you may want to use these trig identities:

$$\sin^2 \theta + \cos^2 \theta = 1$$

$$\sin 2\theta = 2 \sin \theta \cos \theta$$

Problem 5. Acheson, Exercise 5.7.

ME 563 - Intermediate Fluid Dynamics - Su
HW 4 - due 15 March

Problem 1 - Isotropy of the stress tensor. Consider the example given at the end of Lecture 18. Ignore pressure. Compute the forces acting on a square fluid element in both coordinate systems, the original one (\hat{e}_1, \hat{e}_2) and the one rotated by 45° . How do the forces expressed in the different coordinate systems relate? (This is similar to the book example starting on p. 210, but our velocity field is different, and I want you to work out all forces.)

Problem 2. The Burgers vortex.

- (a) Acheson, Exercise 5.19.
- (b) Compute the terms in Eq. 5.38 in the book for the Burgers vortex of part (a) and the decaying line vortex given by Eq. 2.37 in the book. Explain what the terms in 5.38 are describing for both cases.

Problem 3. Acheson, Exercise 5.18.

Problem 4. Acheson, Exercise 6.3.

Problem 5. Acheson, Exercise 6.4.

Problem 6. Acheson, Exercise 6.6.

ME 563 - Intermediate Fluid Dynamics - Su
HW 5 - due 5 April

Problem 1 – Kinetic energy dissipation. Consider an infinitely long cylindrical tube with radius R , inside of which there is a flow of viscous Newtonian fluid. The flow is parallel, so

$$\mathbf{u} = u(r)\hat{e}_z.$$

For a fixed mass flow rate in the tube, \dot{m} , find $u(r)$. Assume that $u(r)$ is of the form $u(r) = A + Br^\alpha$. Do **not** use the Navier-Stokes equations.

Problem 2 – Scalar transport. Let $\zeta(\mathbf{x}, t)$ ('zeta') represent the concentration of a diffusive, conserved scalar. The units of ζ are (1/volume). At any point, the flux of the scalar due to diffusion is $\mathbf{q} = -D\nabla\zeta$, where D is the diffusivity of the scalar. D has units (length²/time) and is a constant. Derive the differential equation describing the time evolution of ζ . As a starting point, you may want to apply control volume principles.

Problem 3 - Acheson, Exercise 8.2.

Problem 4 (20 pts) - Acheson, Exercise 8.4. This is a long problem. Note the following.

1. When Acheson says one of the boundary conditions is “ $u \rightarrow 0$ as we leave the jet,” he means $u \rightarrow 0$ as $y \rightarrow \pm\infty$.
2. In the derivation of Eq. (8.65), keep in mind what symmetry the jet has.
3. In the hints you are told that $g^{1/2}g'$ has to be constant. Why?
4. In solving for f , these will be useful:
 - $\sin^2 + \cos^2 = 1$.
 - $\tanh x = \frac{e^x - e^{-x}}{e^x + e^{-x}}$
 - $\frac{d(\sinh x)}{dx} = \cosh x$, $\frac{d(\cosh x)}{dx} = \sinh x$.
5. Don't use Acheson's reasoning to estimate the width, δ , of the jet. Think instead about what the similarity solution means, and show graphically how the curves of u vs. y differ as x increases, and how they are similar.
6. You don't need to do anything with the material inside the square brackets on p. 295.
7. If you can't solve any part of the problem, just assume the result and pick up the next part (notice this problem is worth 20 points).

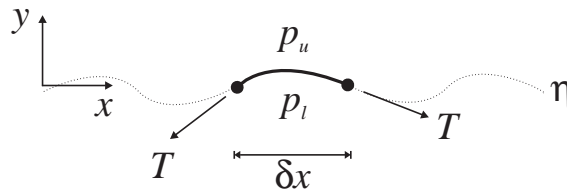
ME 563 - Intermediate Fluid Dynamics - Su
HW 6 - due 26 April

Problem 1 – Harmonic waves. (5 pts) Consider a sinusoidal disturbance defined by

$$\eta(x, t) = A \sin(kx - \omega t). \quad (1)$$

Define the period of this disturbance, T , as the time between the passage of adjacent peaks of the disturbance as seen by an observer at fixed x . Write (1) in terms of T and the wavelength, λ , instead of k and ω .

Problem 2 – Surface tension. (5 pts) Before doing this problem, read §3.4 up to Eq. 3.44, then look at the figure below –



Consider the element of the free surface between the dots, with length δx on the x -axis. The surface tension vectors at the endpoints have magnitude T (which is treated as a constant) and directions as indicated. Show that the vertical component of the surface tension at each endpoint is $T \partial \eta / \partial x$, and then verify the first equation on p. 75 in the text by using Taylor expansions of the terms on the left side. Finally, give the expression relating the pressure above and below the surface element, p_u and p_l . On which side of the surface is the pressure higher?

Problem 3 (10 pts) - Acheson, Exercise 3.2. In this problem you don't have to consider the surface tension between the interfaces.

Problem 4 (10 pts) - Acheson, Exercise 3.6. Here you do need the surface tension, as covered in §3.4. You don't have to do the last part of the problem, deducing the condition for instability.

530.621: Graduate Fluid Mechanics I

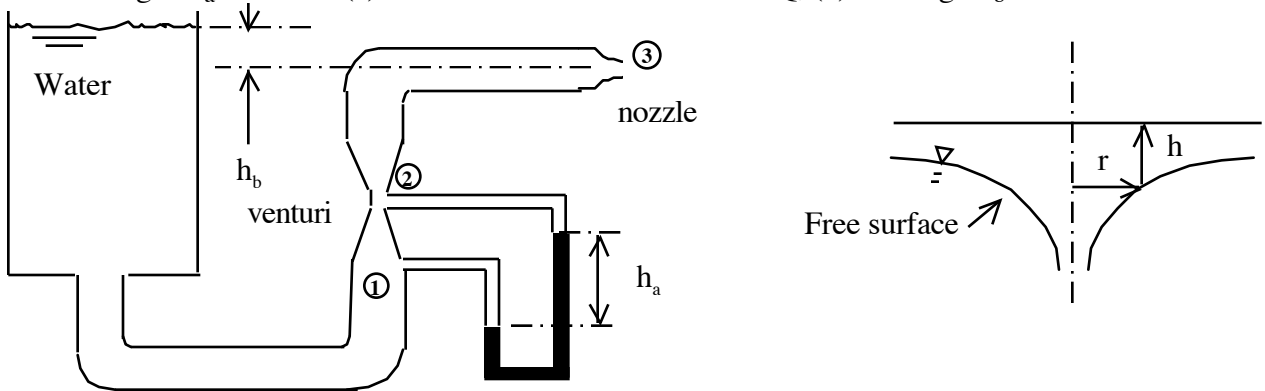
Homework Set 4, due Wednesday Oct 15 with exam

Reading: Panton: Chapter 6, 7.2. Panton does not discuss Bernoulli equation at length, so you should complement this material by reviewing your undergraduate course notes, read other books of the recommended list, etc...

Problems: From Panton: 6.4

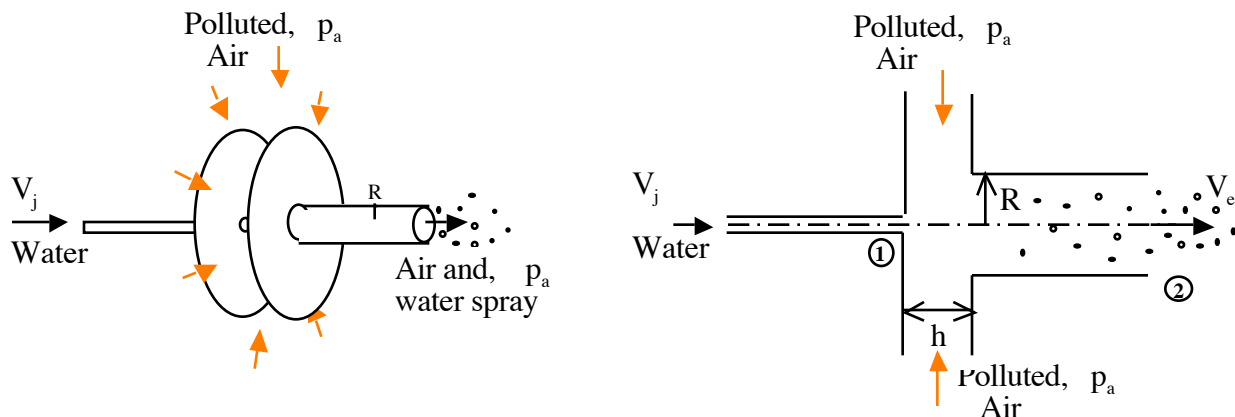
And*:

2. A nozzle is connected to a venturi and to a tank as indicated in the figure. Water flows in a steady fashion from the tank through the nozzle. The cross-sectional areas at indicated sections are $A_1=100 \text{ cm}^2$, $A_2=10 \text{ cm}^2$ and $A_3=8 \text{ cm}^2$. The legs of the mercury manometer are filled with water and the reading is $h_a=300\text{mm}$. (a) Calculate the volume flow rate Q , (b) the height h_b .

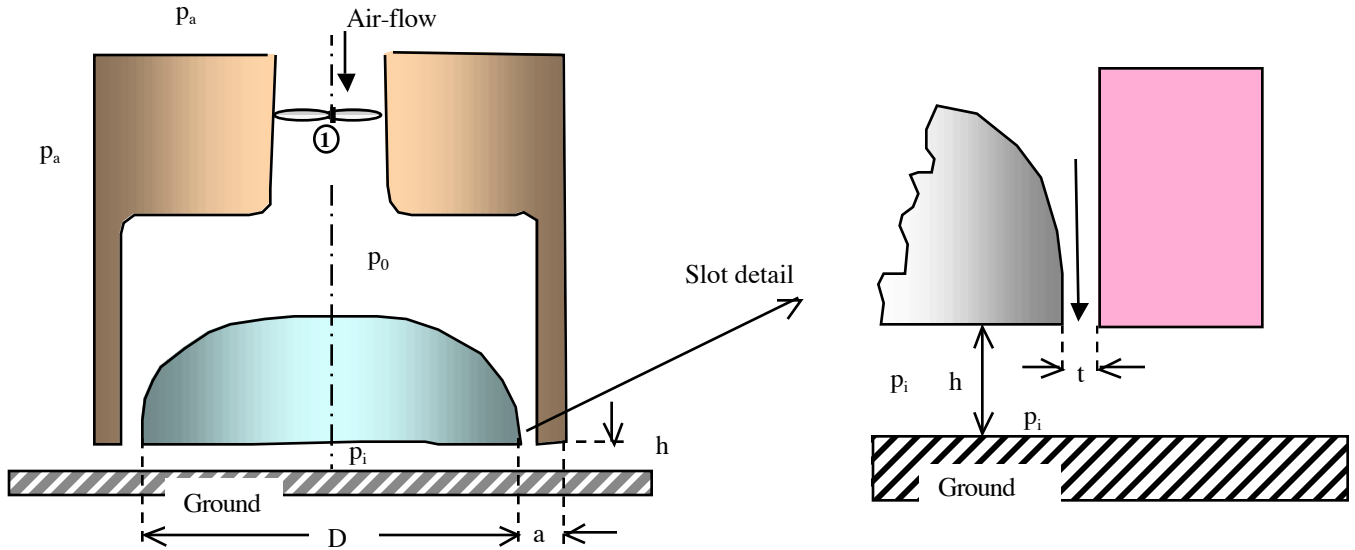


3. Not too close to the center of a whirlpool, the free surface takes on a shape qualitatively shown in the figure. For this region, it is known that particle paths are concentric circles and that the velocity varies with radius r according to $V = \omega/(2\pi r)$. Neglecting friction, determine the shape of the free surface in terms of r , ω and fluid properties. What happens near $r=0$?

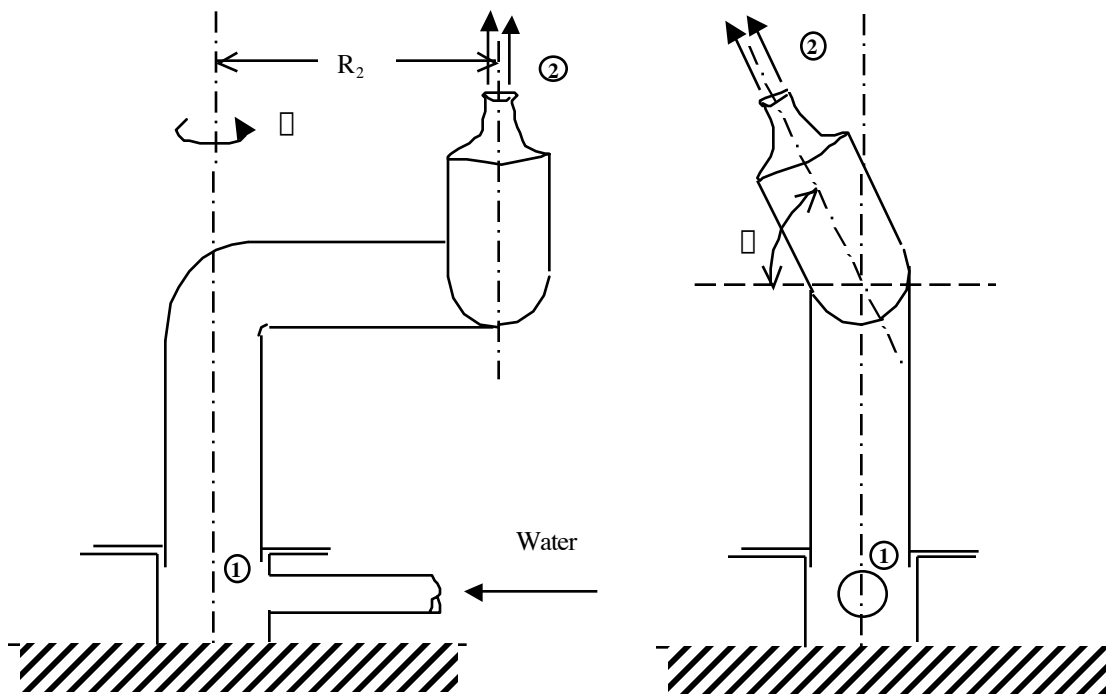
4. A scrubber is a device which removes pollutants from air by allowing the air to pass through a spray of water droplets. The pollutants are absorbed by the droplets, and the air that passes through is clean. An inventor designs the scrubber shown in the figure. A high-speed water jet with velocity V_j and mass flow rate m_w enters at section 1 and quickly breaks up into droplets which become dispersed in the tube. At station 2, the water droplets and the air are moving at essentially the same uniform speed V_e . The device automatically draws air from a source at atmospheric pressure. Calculate the mass flow rate of air, m_{air} .



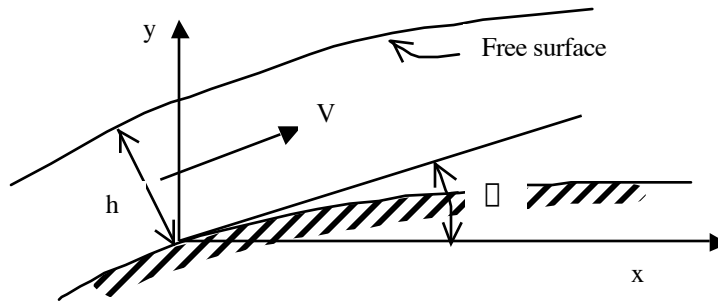
5. A hovercraft is supported by a high-pressure curtain jet. Air is compressed from atmospheric pressure p_a to stagnation pressure p_0 by means of a compressor, and then issues from a narrow slot of width t that extends around the periphery of the base. The whole apparatus has a weight W and floats at a distance h above the ground. Assuming incompressible, inviscid, and steady flow, and that the mean pressure acting on the bottom surface is much smaller than p_0 ($p_i \ll p_0$), and that $t \ll D$ and $t \ll h$, compute h/D in terms of driving pressure $p_0 - p_a$, W , t and D .



6. Consider a lawn sprinkler with radius R_2 . The jet exits at an angle θ to the plane of revolution. At station 1, the pressure is p_1 and the velocity is negligible. In the jet discharge, the pressure p_2 is atmospheric. Neglecting gravity and friction, and assuming that the sprinkler rotates at constant speed, (a) derive an expression for angular velocity in terms of p_1, p_2, θ, ρ and R_2 . (b) Find upwards force R_y exerted by the lower flange on the upper flange near station 1, in terms of p_1, p_2, A_1, A_2 and θ .

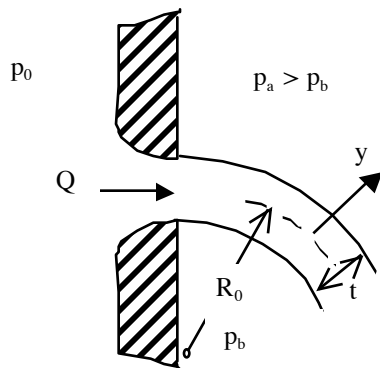


7. Water flows in a 2D channel whose bottom surface is inclined locally at an angle θ . The flow is steady and there is a free surface at the top. Local water velocity is V and local depth of channel is h . The angle θ and its rate of change are small. Show that the slope of the free surface is given by $[F^2/(F^2-1)] \tan\theta$, where $F = V/(gh)^{1/2}$ is the so-called Froude number.



8. An incompressible inviscid fluid escapes through a plane 2D nozzle from a large reservoir where the pressure is p_0 , to a region where the pressure is maintained at the pressure p_a above the jet and at p_b below the jet. The flow is steady and gravity effects are negligible. At some distance from the nozzle, the jet assumes an equilibrium configuration with a thickness t , a radius of curvature R_0 at the center of the jet sheet. Show that the radius R_0 is related to the volume flow rate Q and the pressure drops by

$$R_0 = [(1+N^{-1/2})/\ln(N^{-1})] Q / [2 (p_0-p_b) / \rho]^{1/2}, \quad \text{where } N = (p_0-p_b)/(p_0-p_a).$$



9. Consider a bubble of high-pressure gas expanding in an incompressible liquid in a spherically symmetric fashion. The gas is not soluble in the liquid, and the liquid does not evaporate into the gas. At any instant, $R(t)$ is the radius of the bubble, p_g is the gas pressure, ρ is the liquid density and p_∞ is the liquid pressure at a great distance from the bubble and σ is the surface tension. Neglecting gravity, show that the radius of the bubble is described by the (Rayleigh-Plesset) equation:

$$R \frac{d^2R}{dt^2} + \frac{3}{2} \left[\frac{dR}{dt} \right]^2 + \frac{2\sigma}{\rho R} = (p_g - p_\infty) / \rho.$$

(Hint: First find the water velocity at any radial distance in terms of $R(t)$).

* Problems selected and adapted from 'Advanced Fluid Mech.' course notes (MIT, 1984), courtesy of Prof. O. Knio.

530.621: Graduate Fluid Mechanics I

Homework Set 5, due Monday Nov. 3, 2003

Reading: Panton: Chapter 8

Problems: From Panton: 8.6, 8.15

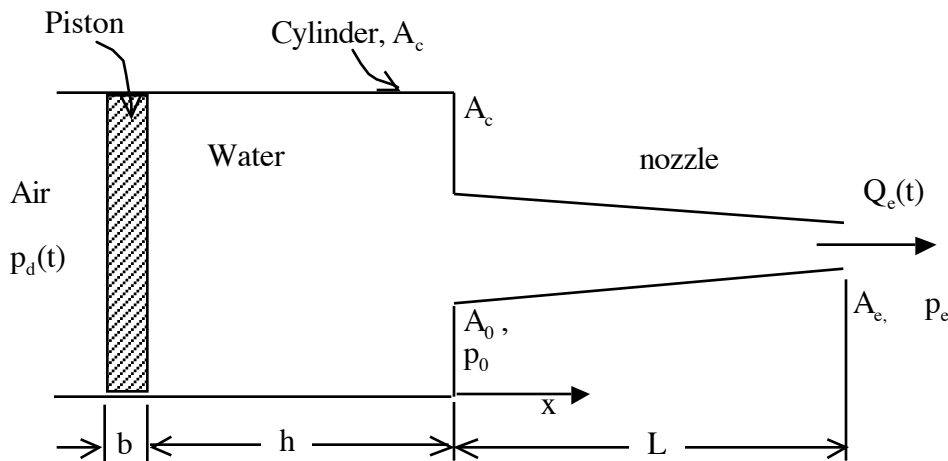
And*:

3. The figure shows a "water cannon" to produce slugs of water at very high speed. Initially, with the nozzle exit temporarily closed by a light "cork", the cylinder and nozzle are loaded with water. Air at high pressure is supplied behind the piston; the cork pops out and a jet of water is accelerated out of the nozzle at a volume flow rate $Q_e(t)$. The cross-sectional area of this particular nozzle follows:

$$A = A_0 - (x/L)(A_0 - A_e).$$

It is agreed to assume that (1) the flow is 1D, (2) water is incompressible, (3) viscous effects are negligible, (4) the cross-sectional area of the cylinder is extremely large compared to that of the nozzle, (5) the exit pressure $p_e = p_{atm}$, (6) $h \ll L$ and $b \ll h$, and (7) the driving pressure difference $p_d(t)$ is known, increasing function of time.

- (a) Calculate $(p_d - p_0)$ by making appropriate approximations.
- (b) Derive the governing equation for Q_e , in terms of $(p_d - p_0)$, L , A_0 , A_e , and relevant fluid properties.
- (c) Assuming $Q_e(t)$ has been determined, find the total tensile force $F(t)$ holding the nozzle to the cylinder.
- (d) Express $F(t)$ and its dependence on relevant parameters in dimensionless form.



4. The velocity of propagation c of a surface wave in deep water is assumed to depend on the wavelength λ , the fluid density ρ , acceleration of gravity g and the surface tension σ . Viscous effects can be ignored.

(a) Using dimensional analysis, obtain a functional relationship for the dimensionless propagation velocity.

(b) Use dimensional analysis to obtain an expression for c assuming that surface tension forces are negligible compared to gravitational forces.

(c) Same, but assuming gravitational forces are negligible compared to surface tension forces.

(d) Formulate a dimensionless group which indicates relative importance of gravity and surface tension effects.

(e) On the basis of (b-d), construct a schematic chart showing the variation of c with λ , all other parameters remaining fixed.

5. It is proposed to evaluate characteristics of a journal bearing by testing a similar model 1/6th the size.

(a) Assuming inertial forces in the oil are negligible (density is irrelevant), find the dimensionless form of an equation connecting the friction torque T , the load supported W , the diameter of the bearing D , the angular speed ω , the oil viscosity μ , and the clearance h (difference in diameter of bearing and shaft).

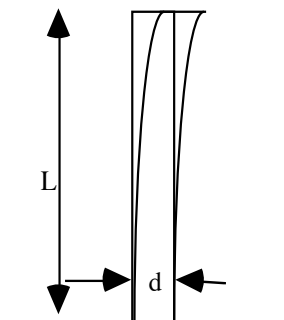
(b) If the prototype is designed to carry a load of 15 tons at 120 rpm, and the model is to be tested at 180 rpm using the same oil as the propotype, what should be the model load?

(c) With the load of part (b), what would be the relationship between the frictional torques in the model and prototype?

6. The vibrations of a long thin cantilevered rod of diameter d and length L are damped because of the drag exerted by the surrounding fluid which has density ρ , viscosity μ , and bulk modulus $\beta = -\rho(dp/d\rho)$. To characterize the decay of periodic motion, we are interested in how the dissipation to the fluid, P_D , depends on the oscillation frequency f , the amplitude of the motion at a particular instant, z , and any pertinent fluid properties.

(a) Define a dimensionless power dissipation and find the dimensionless groups it depends on.

(b) The independent groups you obtained in (a) may or may not have a clear physical meaning. Combine them as necessary to form a set of dimensionless groups which most clearly portray the important physics of the problem. Give a physical interpretation of each.



7. A pair of plates of length L is hinged at O such that the angle $\theta(t)$ contained between the plates may vary in time. An incompressible fluid of density ρ and viscosity μ fills the triangular gap between the plates and the space surrounding the plates. The angle θ is always small, so that $\sin\theta$

$\sim \theta$ and $\cos \theta \sim 1$. When the angular velocity is positive, fluid is pulled into the gap from the surroundings, and vice-versa. The plates are very wide in the direction normal to the paper, and the flow may be considered as two-dimensional. It is agreed that $\partial p / \partial y$ is negligible.

(a) Express the pressure difference $p(x) - p(L)$ in terms of dimensionless parameters. It is assumed that $\theta(t)$ is a known function of time. Explain the reason for including each independent variable you choose. In particular, explain whether or not the successive time derivatives $d\theta/dt$, $d^2\theta/dt^2$ etc. should be included.

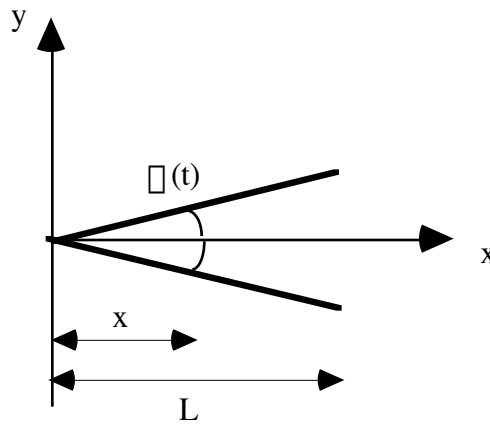
(b) Let $U(x,t)$ be the average velocity in the x -direction, at the location x and time t . Solve (dimensionally) for U as function of θ , $d\theta/dt$ and x .

(c) Show that the order-of-magnitude estimate $v/u \sim O(\theta)$,

where v is the local vertical velocity and u the local horizontal velocity in the x -direction, holds.

(d) From an order-of-magnitude analysis of the Navier-Stokes equation in the x -direction, determine the criteria which would allow one to consider the local flow as inertia-free.

(e) What are the criteria for the entire flow to be inertia-free? Show that even if the entire flow cannot be considered inertia-free, at least some part of it may be so considered.



8. An engineer tests an aircraft of length 20m and cross-sectional area 6m². He tests a scale model of length 1.2m in a water tunnel. It is found that the drag force of the model at cruising conditions depends on the flow speed according to

$$D = 20V + 0.4 V^2 \text{ (N)}, \text{ where } V \text{ is in m/s.}$$

It the test, the model stalls when V decreases to 10 m/s.

(a) Predict the stalling speed of the real aircraft.

(b) Obtain an expression for the drag force on the real aircraft in terms of its flying speed.

(c) Calculate the power output required by the real aircraft to maintain a steady cruising speed of 100 m/s.

* Problems selected and adapted from 'Advanced Fluid Mech.' course notes (MIT, 1984), courtesy of Prof. O. Knio.

530.621: Graduate Fluid Mechanics I

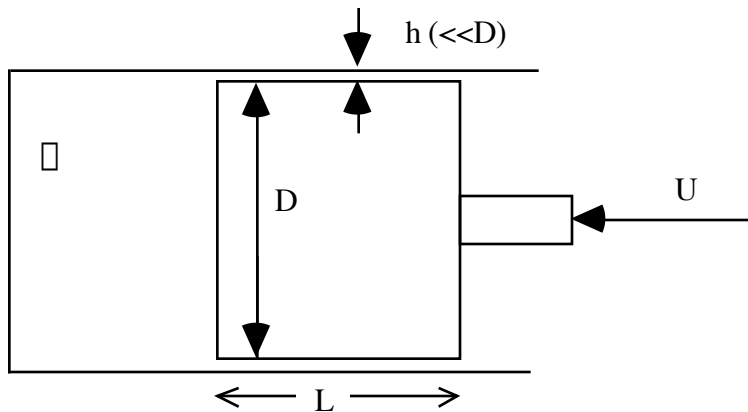
Homework Set 6, due Monday Nov 10.

Reading: Panton: Chapter 7 and 11 (in depth reading)

Problems: From Panton: This is like a single problem: 7.5,7.6,7.7,7.8 (they all refer to 7.5, not 7.4).

And:

4. A piston moves in a cylinder whose dimensions are shown below. The piston moves in at a speed U . Assume that the gap is much smaller than the piston length. Solve for the velocity profile in the gap. Find the force that must be applied to the piston.

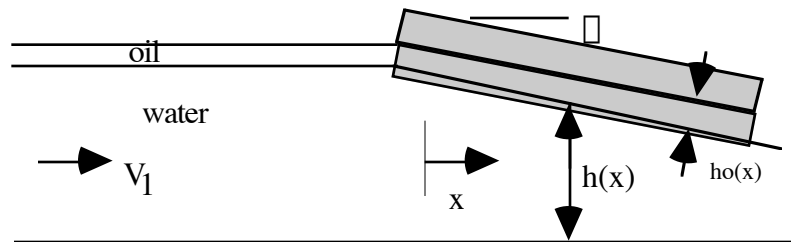


5.* A horizontal water stream covered with a very thin layer of oil impinges on a flat plate which is inclined at an angle α with the horizontal direction. Upstream, the water depth is h_{w1} and the oil film has depth h_{o1} ($h_{o1} \ll h_{w1}$). The two fluids move at the same speed V_1 . The water density is larger than that of oil ($\rho_w > \rho_o$). The plate elevation is given by $h(x) = h_1 - \alpha x$. The oil thickness is $h_o(x) \ll h(x)$. Viscous forces dominate in the oil layer, but the water stream may be considered inviscid throughout. We may assume 2-D, steady flow, neglect gravity and end-effects near edges of plate.

(a) If the water flow is inviscid, what force drives the thin film forward?

(b) Find the pressure distribution $p(x)$ in the water

(c) Find the oil layer thickness $h_o(x)$ on the plate.



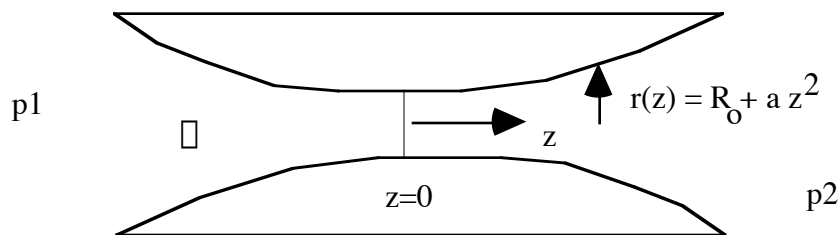
* Problem adapted from 'Advanced Fluid Mech.' course notes (MIT, 1984), courtesy of Prof. O. Knio.

530.621: Graduate Fluid Mechanics I
Homework Set 7, due Wednesday Nov. 19

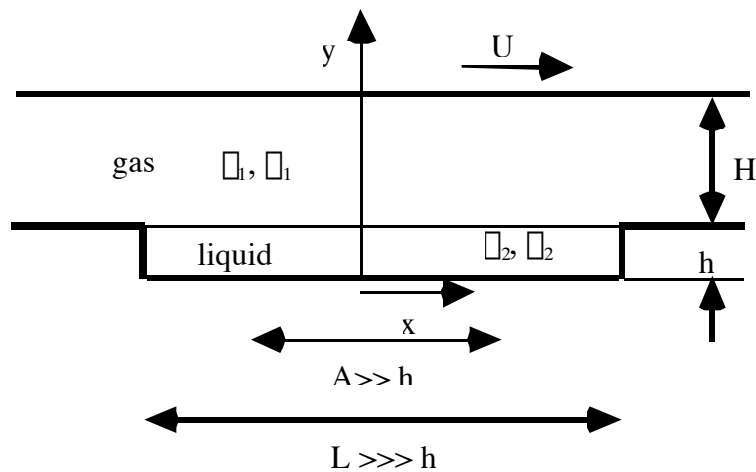
Reading: Panton: Chapter 21

Problems:

1. Find the pressure difference $p_1 - p_2$ required to drive a volume flow-rate Q across the axisymmetric channel with parabolic shape as shown. Use the lubrication approximation. Express limits of validity of the approach in terms of given variables.



2. (previous midterm #2 problem): Consider two-dimensional laminar flow of Newtonian gas (density ρ_1 , viscosity μ_1) in a channel, that has a groove with Newtonian liquid. Gas and liquid are separated by a horizontal interface. The upper wall of the channel moves from left to right at speed U . The lower wall contains a very long (in x and z directions) and narrow (in y -direction) groove filled with a liquid of density ρ_2 , viscosity μ_2 . H and h are of the same order of magnitude. In the central region A away from the ends of the groove, the streamlines are essentially horizontal ($v=0$).



(a) (5 pt) Assuming that the upper wall has been in motion for a very long time, sketch the streamlines in the gas and in the liquid region.

(b) (10 pt) Continue assuming that in region A, the flow can be considered as horizontal ($v=0$) and steady, write down the equations governing the motion in the gas (velocity $u_g(y)$) and in the liquid (velocity $u_l(y)$). In this problem, you may assume that, due to very high surface tension, the interface remains horizontal as shown above, even when a pressure difference develops between liquid and gas, and between left and right edges of the groove.

(c) (5pt) Write down all necessary boundary conditions.

(d) (5pt) Write down the appropriate condition from which you can solve for the horizontal pressure gradient $\partial p/\partial x$ that exists in the liquid groove.

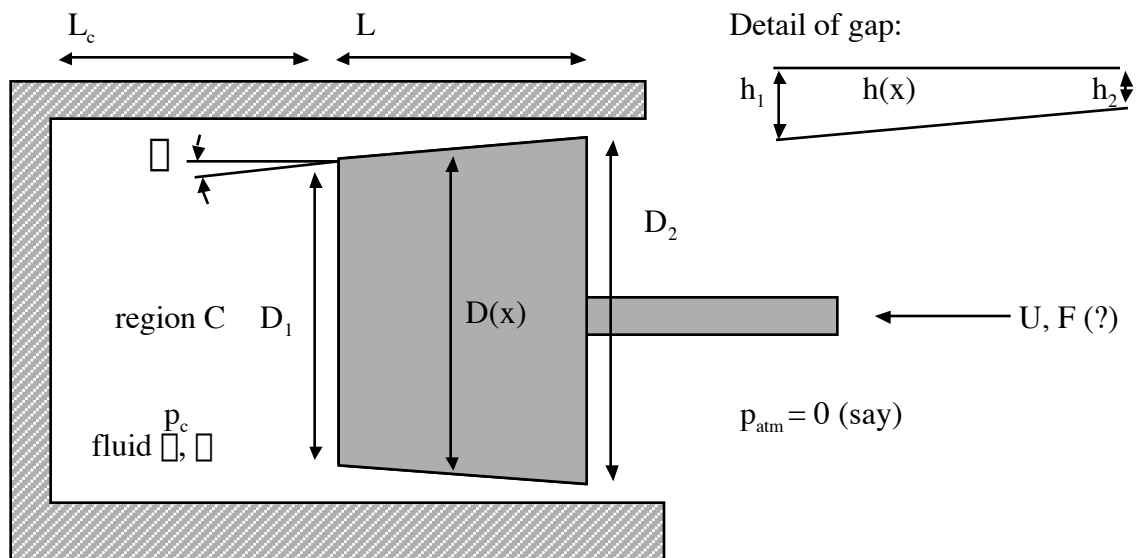
(e) (5pt) Solve for $u_g(y)$, $u_l(y)$, and $\partial p/\partial x$.

(f) (5pt) If at $t < 0$ the upper wall and fluids were stationary, and the upper wall impulsively starts moving at speed $U=0.1$ m/s at $t=0$, estimate (no exact solution required) the order of magnitude of the time required before the gas near the liquid surface ($y=h$) achieves a significant fraction of its final velocity. Other numerical parameters are as follows: $h=1$ cm, $H=2$ cm, $\nu_l=1 \cdot 10^{-4}$ m²/s.

(g) (5pt) Still assuming fully-developed conditions in the x-direction and a flat interface at $y=h$, clearly write out (do not solve) the complete mathematical formulation which would allow you to solve for unsteady velocity profiles in both fluids. (general notation, do not use numerical parameters of part (f).)

3. (previous exam problem) A piston with a conical shape (segment of a cone) is pushed into a liquid-filled cylindrical container as shown in the figure. The incompressible fluid inside region C is squeezed through a narrow passage of linearly decreasing height. We are interested in calculating the force required to push the piston at a constant speed U . The analysis will be based on the following assumptions:

- (i) The angle α (approximately equal to $(h_1-h_2)/L$) is very small (but not negligible).
- (ii) The gap height h_1 is much smaller than D_1 . Also, $h_1 \ll L_c$.



(a) (4p) Without solving partial differential equations, calculate the average horizontal velocity (in the gap) of the fluid which is being squeezed out, in terms of U , D_1 , h_1 .

(b) (4p) Give an estimate of the ratio of inertia and viscous terms in the momentum (NS) equation in the horizontal direction for the flow in the gap. In terms of μ , U , D_1 , h_1 and fluid properties, under what condition can we neglect the inertia terms in the equations?

Now, assume that this condition is met. Therefore, we can assume mixed Poiseuille+Couette flow at any location x in the gap, but with a varying (unknown) pressure gradient (lubrication approximation).

(c) (4p) Is it important to keep the moving piston wall (Couette part) when solving for the velocity profile inside the gap? To answer this question, construct a dimensionless parameter β (velocity U divided by mean velocity in gap). Consider the limit when $h/D_1 \rightarrow 0$.

(d) (12p) Neglecting the Couette part in the flow inside the gap, find the pressure in region C. (I am assuming you are able to integrate $(a+bx)^{-2}$).

(e) (12p) Find the force F . (I am assuming you are able to integrate $(a+bx)^{-1}$). There are three basic contributions to the force. Compute each, and determine which is dominant (assume $L \sim O(D)$). When integrating, you may also assume that $D(x) \sim D_1$.

4. Solve axisymmetric Stokes flow of a liquid spherical droplet of viscosity μ_i and radius a , in another external fluid with viscosity μ_e . Some results are exhibited in Panton, but you are asked to derive and show all details, following approach used in class.

(a) Using the general expression derived in class (which is valid for both external flow and internal circulation) you must now solve for 8 coefficients $A_i, A_e, B_i, B_e, \dots$ using appropriate boundary conditions. Obtain the velocity field u_r and u_θ both inside and outside the sphere in terms of the parameter $\beta = \mu_e/\mu_i$.

(b) Find the pressure distribution at $r=a$.

(c) Find the distribution of tangential and normal viscous stresses (note: now the normal viscous stress need not be zero)

(d) Integrate to find the drag force in terms of U , μ_e , a and β

(e) Confirm that you recover the case of a solid sphere if $\beta \rightarrow 0$. Obtain an expression for drag force for the case of a bubble in a liquid of viscosity μ_e , by letting $\mu_e/\mu_i \rightarrow \infty$.

5. Using any computer plotting package, plot the streamlines for Oseen's solution of low Reynolds number flow over a sphere in a frame of reference moving with the sphere. Consider $Re=0.001$, $Re=0.1$ and $Re=1$. Comment on qualitative differences (e.g. front-aft symmetry, etc.).

530.621: Graduate Fluid Mechanics I

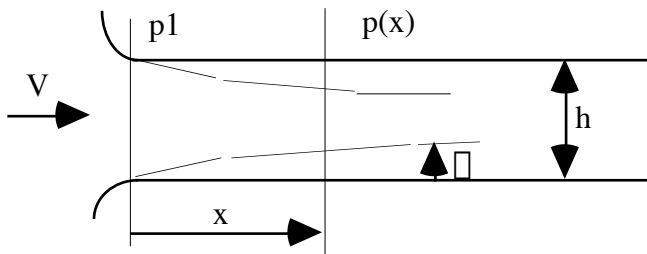
Homework Set 8

Due by 5pm Monday Dec 8 (in mailbox box Latrobe 127)

Reading: Panton: Chapter 16.4, all of chapter 20.

Problems: From Panton: 20.3, 20.6.

3. Consider the steady laminar incompressible flow in entry of a 2D channel of constant cross-section. Boundary layers on the walls begin at $x=0$ and it is agreed to assume that the velocity profile in the boundary layer is linear with distance from the wall. Derive a formula relating the pressure drop coefficient $[p(x) - p_1] / (\rho V^2) (h/x)$ to the length and breadth Reynolds numbers, defined respectively as $\rho V x / \mu$ and $\rho V h / \mu$.



4. A 2-D channel has a converging-diverging cross-section, whose height is given by $h(x) = h_0$ for $x < 0$ and $x > L$, and $h(x) = h_0 [1 + \epsilon (1 - \cos(2\pi x/L))]^{-1}$ for $0 < x < L$. The bottom wall is straight and only the top wall has the cos-shape. Inlet velocity is U_0 . To a first approximation, neglect the effect of displacement thickness on the outer flow. A laminar boundary layer has been developing on the bottom surface. Assume that at $x=0$, the displacement boundary-layer thickness is $\delta^*(0)$.

(a) Using the momentum integral formulation with an appropriate parabolic test-profile between $y=0$ and $y=\delta^*(x)$, write down an equation for $\delta^*(x)$.

(b) Using the values $U_0=1$ m/s, $\nu=10^{-5}$ m²/s, $h_0=0.03$ m, $\epsilon=0.2$, $L=0.3$ m and $\delta^*(0)=1$ mm, solve this ODE numerically (by writing a short program). Integrate between $x=0$ and $x=L$ using a simple first-order Euler integration:

$$d\delta^*/dx = g(\delta^*, x) \text{ leading to } \delta^*_{i+1} = \delta^*_i + (x_{i+1} - x_i) g(\delta^*_i, x_i),$$

starting from the initial (boundary) condition at $i=0$, up to $i=100$. $x_i = (i-1) L / 100$.

Plot the resulting $\delta^*(x)$ in dimensional units.

50 points

Problem 1 - Short answers. (Total 20 pts)

(a) (5 pts) Consider the velocity field defined in Cartesian coordinates as:

$$\mathbf{u}(x, y, z) = \left(x^2y - \frac{1}{2}yz^2, -\frac{1}{2}xy^2 - \frac{1}{2}xz^2, -xyz \right).$$

Is it possible to write a velocity potential, ϕ , for this flow? Why or why not? (Even if it is possible, you don't need to find ϕ .)

\exists VELOCITY POTENTIAL ϕ IF $\nabla \times \vec{u} = 0$ (3)

$$\nabla \times \vec{u} = \begin{vmatrix} \hat{e}_x & \hat{e}_y & \hat{e}_z \\ \frac{\partial}{\partial x} & \frac{\partial}{\partial y} & \frac{\partial}{\partial z} \\ u & v & w \end{vmatrix} = \hat{e}_x \underbrace{\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z} \right)}_{\omega_x} + \hat{e}_y \underbrace{\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x} \right)}_{\omega_y} + \hat{e}_z \underbrace{\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right)}_{\omega_z}$$

$$\omega_x = \omega_y = 0$$

$$\omega_z = -\frac{1}{2}y^2 - \frac{1}{2}z^2 - x^2 + \frac{1}{2}z^2 = -\frac{1}{2}y^2 - x^2 \neq 0$$
 (2)

IT IS NOT POSSIBLE TO WRITE A VELOCITY POTENTIAL.

(b) (5 pts) Draw the streamline patterns for the following stream functions: (1) $\psi = 2r^3$, (2) $\psi = 4\pi/\sqrt{r}$. Think before you compute any derivatives.

STREAMLINES ARE LINES OF CONSTANT ψ (3)

FOR BOTH (1) AND (2), LINES OF CONSTANT ψ

ARE LINES OF CONSTANT $r \rightarrow$

LINES OF CONSTANT r ARE CIRCLES CENTERED AT

THE ORIGIN \rightarrow STREAMLINES ARE CIRCLES. (2)



(IF MENTION $u_r = \frac{1}{r} \frac{\partial \psi}{\partial \theta}$, $u_\theta = -\frac{\partial \psi}{\partial r}$: (1 PT)) (MENTION $u_r = 0$: (2 PTS))

- (c) (10 pts) Consider the discussion of D'Alembert's paradox in §4.13 in the book, which makes use of the figure below. A body sits in a flow of ideal fluid in a long, straight channel with uniform cross-section – the paradox is that it feels no drag force, D . The discussion in the text hinges on the assumptions that the flow is uniform and has speed U_0 both far upstream and *and* far downstream of the body. The assumption that the flow is uniform far upstream is easily justified as part of the definition of the problem. It is also easy to justify that the flow is parallel (i.e. $\mathbf{u} = (u, 0, 0)$) far upstream and downstream, since we've assumed that the channel has a uniform cross-section. However, can you think of a way to justify the assumption that the flow is *uniform* far *downstream* of the body?

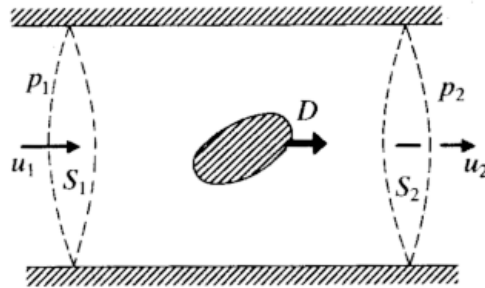


Figure 1: Sketch for illustrating D'Alembert's paradox.

- UNIFORM FLOW UPSTREAM \Rightarrow IRROTATIONAL FLOW FAR UPSTREAM. (4)
- (4) BY CAUCHY-LAGRANGE THEOREM, IRROTATIONAL FLOW OF IDEAL FLUID WILL ALWAYS BE IRROTATIONAL \rightarrow IN PARTICULAR, FAR DOWNSTREAM. PARALLEL, IRROTATIONAL FLOW MUST BE UNIFORM ($\frac{du}{dy} = \frac{dv}{dz} = 0$). (2)
- THE FLOW IS UNIFORM FAR DOWNSTREAM OF THE BODY.

\rightarrow REFERENCE TO BERNOULLI THEOREMS ONLY: (3 PTS)

\rightarrow REFERENCE TO NO VISCOUS SHEAR TERMS: (3 PTS)

PROBLEM 2 CONTINUED

$$\boxed{u_2} \frac{P}{2\mu_2} h^2 + C_2 h + D = 0 \Rightarrow C_2 = -\frac{D}{h} - \frac{P}{2\mu_2} \frac{h^2}{h}$$

$$C_2 = \frac{\mu_1}{\mu_2} C_1 = \frac{P}{2\mu_1} h - C_1 - \frac{P}{2\mu_2} h$$

* LAST PAGE
USING ***

$$C_1 \left[\frac{\mu_1 + \mu_2}{\mu_2} \right] = \frac{Ph}{2} \left[\frac{1}{\mu_1} - \frac{1}{\mu_2} \right] = \frac{Ph}{2} \left[\frac{\mu_2 - \mu_1}{\mu_1 \mu_2} \right]$$

$$\Rightarrow C_1 = \frac{Ph}{2\mu_1} \left[\frac{\mu_2 - \mu_1}{\mu_1 + \mu_2} \right]$$

$$\Rightarrow u_1 = \frac{P}{2\mu_1} \left[(y^2 - h^2) + \left[\frac{\mu_2 - \mu_1}{\mu_1 + \mu_2} \right] h(y+h) \right]$$

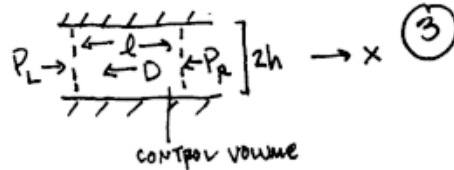
From ***: $D = -\frac{P}{2\mu_2} h^2 - C_2 h$

③

$$\Rightarrow u_2 = \frac{P}{2\mu_2} (y^2 - h^2) + C_2 (y-h), \quad C_2 = \frac{\mu_1}{\mu_2} C_1$$

$$\Rightarrow u_2 = \frac{P}{2\mu_2} \left[(y^2 - h^2) + \left[\frac{\mu_2 - \mu_1}{\mu_1 + \mu_2} \right] h(y-h) \right]$$

b) FOR FORCE EXERTED BY PLATES:



③

Flow is steady, so drag force D balances pressure forces

$$\frac{dp}{dx} = \text{const.}, \text{ so } P_R = P_L + l \cdot \frac{dp}{dx} \quad (1)$$

$\frac{dp}{dx} < 0$

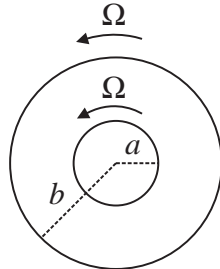
$$\text{NET PRESSURE FORCE}_x = (P_L - P_R) \cdot 2h = -l \frac{dp}{dx} \cdot 2h$$

$$= 2h \left| \frac{dp}{dx} \right| \text{ in } +x\text{-direction} \quad (1)$$

$$= \text{DRAG FORCE EXERTED BY PLATES IN } -x\text{-direction.}$$

↳ (per unit span)

Problem 3. (Total 15 pts) Consider two infinitely long, concentric circular cylinders, of radii a (the inner cylinder) and b (the outer cylinder). The space between the cylinders is filled with a viscous, incompressible fluid with density ρ and viscosity μ . Suppose both cylinders rotate with angular velocity Ω (i.e. the linear velocity at the surface of the inner cylinder is Ωa , and at the surface of the outer cylinder is Ωb .) **Suppose also that the inner cylinder is moving in the out-of-page, z -direction at a constant velocity U .**



The relevant Navier-Stokes equations are (2.22 in the book):

$$\begin{aligned} \frac{\partial u_r}{\partial t} + (\mathbf{u} \cdot \nabla)u_r - \frac{u_\theta^2}{r} &= -\frac{1}{\rho} \frac{\partial p}{\partial r} + \nu \left(\nabla^2 u_r - \frac{u_r}{r^2} - \frac{2}{r^2} \frac{\partial u_\theta}{\partial \theta} \right) \\ \frac{\partial u_\theta}{\partial t} + (\mathbf{u} \cdot \nabla)u_\theta + \frac{u_r u_\theta}{r} &= -\frac{1}{\rho r} \frac{\partial p}{\partial \theta} + \nu \left(\nabla^2 u_\theta + \frac{2}{r^2} \frac{\partial u_r}{\partial \theta} - \frac{u_\theta}{r^2} \right) \\ \frac{\partial u_z}{\partial t} + (\mathbf{u} \cdot \nabla)u_z &= -\frac{1}{\rho} \frac{\partial p}{\partial z} + \nu \nabla^2 u_z \end{aligned}$$

where we also have

$$\begin{aligned} (\mathbf{u} \cdot \nabla) &= u_r \frac{\partial}{\partial r} + \frac{u_\theta}{r} \frac{\partial}{\partial \theta} + u_z \frac{\partial}{\partial z}, \quad \text{and} \\ \nabla^2 &= \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2}{\partial \theta^2} + \frac{\partial^2}{\partial z^2}. \end{aligned}$$

- a) (7 pts) Assuming steady flow, find the velocity field between the cylinders. There is no pressure gradient $\partial p / \partial z$ and you can ignore gravity. Assume that the velocity is of the form $\mathbf{u} = u_\theta(r)\hat{e}_\theta + u_z(r)\hat{e}_z$. The book example on page 44 is useful.
- b) (8 pts) Suppose the pressure at the surface of the inner cylinder is p_a . Find the pressure field between the cylinders.

(a) • WRITE THE u_θ EQUATION FOR $\vec{u} = u_\theta(r)\hat{e}_\theta + u_z(r)\hat{e}_z$: $\frac{\partial p}{\partial \theta} = 0$ BY ARGUMENT ON P. 44

$$\begin{aligned} \Rightarrow 0 &= \nu \left(\frac{1}{r} \frac{d}{dr} \left(r \frac{du_\theta}{dr} \right) - \frac{u_\theta}{r^2} \right) \\ &= \frac{d^2 u_\theta}{dr^2} + \frac{1}{r} \frac{du_\theta}{dr} - \frac{u_\theta}{r^2} \quad \text{(1) SAME EQUATION AS EXAMPLE P. 44} \end{aligned}$$

$$\Rightarrow u_\theta = A r + \frac{B}{r} ; A = \frac{\Omega_2 b^2 - \Omega_1 a^2}{b^2 - a^2}, B = \frac{(\Omega_1 - \Omega_2) a^2 b^2}{b^2 - a^2} ; \Omega_2 = \Omega_1 = \Omega,$$

so $u_\theta = \Omega r$ (2) (SOLID BODY ROTATION)

(More space for Problem 3)

- EQUATION FOR u_z : USE $\frac{\partial P}{\partial z} = 0$

$$0 = \nu \left(\frac{\partial^2 u_z}{\partial r^2} + \frac{1}{r} \frac{\partial u_z}{\partial r} \right) \Rightarrow \frac{\partial^2 u_z}{\partial r^2} = -\frac{1}{r} \frac{\partial u_z}{\partial r} \quad (1)$$

$$\rightarrow \text{LET } f = \frac{\partial u_z}{\partial r} \Rightarrow \frac{df}{dr} = -\frac{1}{r} f \Rightarrow \frac{df}{f} = -\frac{dr}{r}$$

$$\Rightarrow \ln f = -\ln r + C \Rightarrow (\text{EXPONENTIAL OF BOTH SIDES}) \quad f = \frac{e^C}{r} = \frac{D}{r}$$

$$\rightarrow \frac{\partial u_z}{\partial r} = \frac{D}{r} \Rightarrow u_z = D \ln r + E \quad (1)$$

BOUNDARY CONDITIONS: $u_z(a) = U$, $u_z(b) = 0$

$$u_z(b) = D \ln b + E = 0 \Rightarrow u_z = D(\ln r - \ln b) = D \ln\left(\frac{r}{b}\right)$$

$$u_z(a) = U = D \ln \frac{a}{b} \Rightarrow D = \frac{U}{\ln(a/b)}$$

$$u_z = \frac{U}{\ln(a/b)} \ln r/b \quad (1)$$

~~~~~

- (b) FOR PRESSURE:  $u_r$  EQUATION GIVES  $-\frac{u_\theta^2}{r} = -\frac{1}{\rho} \frac{\partial P}{\partial r} \quad (3)$

$$\frac{dP}{dr} = \frac{\rho u_\theta^2}{r} = \rho \Omega^2 r$$

$$P = \frac{\rho \Omega^2}{2} r^2 + C \quad (3)$$

$$\text{B.C. } P(r=a) = P_a = \frac{\rho \Omega^2 a^2}{2} + C$$

$$\Rightarrow C = P_a - \frac{\rho \Omega^2 a^2}{2}$$

$$P = \frac{\rho \Omega^2}{2} (r^2 - a^2) + P_a \quad (2)$$

~~~~~

Bonus questions from the world of the arts.

1. Which Mahler symphony uses sleigh bells?
 - a) 2nd
 - b) 4th
 - c) 9th
 - d) 256th

2. Who of the following is not an Impressionist (could be more than one)?
 - a) Vincent van Gogh
 - b) Jackson Pollock
 - c) Claude Monet
 - d) Anselm Kiefer

3. The Beach Boys' hit records from the 1960s were originally in mono, not stereo, because:
 - a) Brian Wilson was deaf in one ear
 - b) Stereo recording equipment was expensive in those days
 - c) Capitol Records thought stereo was a fad
 - d) They *were* originally in stereo, what are you talking about?

4. The first line of *Moby-Dick* is:
 - a) "It was a dark and stormy night."
 - b) "It was the best of times, it was the worst of times."
 - c) "'Thar she blows!' "
 - d) "Call me Ishmael."

5. My favorite color is:
 - a) Black
 - b) Gray
 - c) Blue
 - d) None, I'm colorblind

Problem 1 - Short answers. (Total 20 pts)

(a) (10 pts) For a Newtonian fluid, the stress tensor is given by

$$T_{ij} = -p \delta_{ij} + \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right),$$

and the momentum equation (in index notation, and ignoring gravity) is

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j},$$

and the incompressibility condition is

$$\frac{\partial u_j}{\partial x_j} = 0.$$

Now suppose we have a fluid (non-Newtonian) with stress tensor given by

$$T_{ij} = -p \delta_{ij} + \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \gamma u_i u_j.$$

(γ is a constant.) Write the momentum equation (ignoring gravity) and incompressibility condition in index notation for this fluid.

- Incompressibility condition: $\frac{\partial u_j}{\partial x_j} = 0$ (unchanged: has nothing to do with stress tensor) (3)
 - Momentum equation: use Cauchy eq. $\rho \frac{D u_i}{D t} = \frac{\partial T_{ij}}{\partial x_j} + \rho g_i$ (2)
 IGNORE GRAVITY
- (1) $\frac{\partial T_{ij}}{\partial x_j} = \frac{\partial}{\partial x_j} \left[-p \delta_{ij} + \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \frac{\partial}{\partial x_j} \gamma u_i u_j$
 SAME AS FOR NEWTONIAN FLUID \leftarrow (2)
- $= -\frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + \gamma \left[\frac{\partial u_i}{\partial x_j} \cdot u_j + u_i \frac{\partial u_j^2}{\partial x_j} \right]$
- $\Rightarrow \frac{D u_i}{D t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + \frac{\gamma}{\rho} u_j \frac{\partial u_i}{\partial x_j}$ (2)

(b) (10 pts) In working the problem of the steady, two-dimensional flat plate boundary layer, we said that the (inviscid) main flow velocity, $U(x)$, was constant, so that the pressure gradient was zero, i.e. $dp/dx = 0$. Then, we found the velocity in the boundary layer subject to the no-slip condition. But consider the following points:

- For the main flow velocity to be constant despite no driving pressure gradient, it seems that there must be no friction losses in the system.
- But there *is* friction, because the shear stress at the surface of the flat plate is nonzero.

It appears that these two points are inconsistent with each other. Are they? Why or why not?

THERE IS NO INCONSISTENCY⁽³⁾ — THE FRICTION LOSSES (= REDUCTION IN X-MOMENTUM)⁽⁴⁾ ARE ENTIRELY IN THE VISCOUS BOUNDARY LAYER. THE INVISCID MAIN FLOW FEELS NO EFFECT OF DRAG⁽³⁾

(AS THE B.L. PROCEEDS FURTHER ALONG THE FLAT PLATE, THE TOTAL DRAG INCREASES AS MORE BOUNDARY AREA IS INTRODUCED. THIS MEANS MORE X-MOMENTUM IS 'TAKEN' FROM THE FLUID FLOW, WHICH IS MANIFESTED IN THE THICKENING OF THE B.L.)

Problem 2 - Boundary layers. (30 pts)

- a) (5 pts) We've seen how to find velocity fields by minimizing the kinetic energy dissipation. Why can't we use this approach for boundary layers?

THE MINIMUM DISSIPATION FORMULATION ASSUMES THAT INERTIAL FORCES ARE NEGLIGIBLE COMPARED WITH VISCOUS FORCES. BUT IN BOUNDARY LAYERS, WE ASSUME THEY ARE COMPARABLE.

e.g. STEADY, 2-D B.L. EQS.

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{dp}{dx} + \nu \frac{\partial^2 u}{\partial y^2}$$

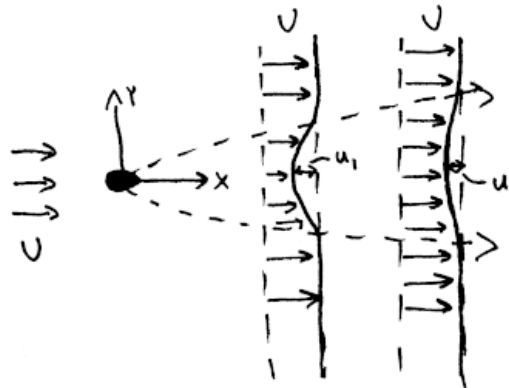
INERTIAL TERMS
← ASSUMED TO BE OF SAME ORDER →
VISCOUS TERM

- b) (3 pts) (The rest of this problem is based on Acheson's Exercise 8.5). Consider the flow downstream of a two-dimensional streamlined body at high Reynolds number. The flow upstream of the body is constant and in the x -direction, i.e. $\mathbf{u} = (U, 0)$; assume that the body is centered at $y = 0$ and is symmetric about $y = 0$. There is a thin wake downstream of the body in which the velocity varies much more rapidly with y than with x . Assume that we are far enough downstream of the body that we can write the velocity field as

$$\mathbf{u} = (U + u_1(x, y), v_1(x, y)),$$

where $u_1 < 0$, $|u_1/U| = O(\epsilon)$, and $\epsilon \ll 1$.

First, draw what the u vs. y profiles might look like for a couple of different positions downstream of the body.



- c) (7 pts) Let δ be the characteristic width of the wake (in the y -direction) and L be a characteristic length (in the x -direction). Show by explicitly estimating the magnitudes of the terms in the boundary layer equations that the momentum equation reduces to

$$U \frac{\partial u_1}{\partial x} = \nu \frac{\partial^2 u_1}{\partial y^2}.$$

Can you relate δ and L ?

• $\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = \frac{\partial u_1}{\partial x} + \frac{\partial v_1}{\partial y} = 0$. $\left| \frac{\partial u_1}{\partial x} \right| \approx \left| \frac{\partial v_1}{\partial y} \right|$. Use $|u_1| = O(\epsilon U)$
 $\Rightarrow O\left(\frac{\epsilon U}{L}\right) \approx O\left(\frac{|v_1|}{\delta}\right) \Rightarrow |v_1| = O\left(\frac{\epsilon U \delta}{L}\right)$

• $u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial^2 p}{\partial x^2} + \nu \frac{\partial^2 u}{\partial y^2} \rightarrow (U+u_1) \frac{\partial u_1}{\partial x} + v_1 \frac{\partial u_1}{\partial y} = \nu \frac{\partial^2 u_1}{\partial y^2}$

Use $\left| U \frac{\partial u_1}{\partial x} \right| = O\left(\frac{\epsilon U^2}{L}\right)$, $\left| u_1 \frac{\partial u_1}{\partial x} \right| = O\left(\frac{\epsilon^2 U^2}{L}\right)$, $\left| v_1 \frac{\partial u_1}{\partial y} \right| = O\left(\frac{\epsilon^2 U^2}{L}\right)$, $\left| \frac{\partial^2 u_1}{\partial y^2} \right| = O\left(\frac{\epsilon U}{\delta^2}\right)$

TERMS $w/\epsilon^2 \ll$ TERMS $w/\epsilon \Rightarrow U \frac{\partial u_1}{\partial x} = \nu \frac{\partial^2 u_1}{\partial y^2}$

Also $\delta \ll L$; BUT BY COMPARING THESE, ALSO GET $\frac{\delta}{L} = O\left(\sqrt{\frac{\nu}{UL}}\right)$.

- d) (5 pts) Show that

$$\int_{-\infty}^{\infty} u_1 dy = \text{const.} \quad (1)$$

INTEGRATE ABOVE MOMENTUM EQ.:

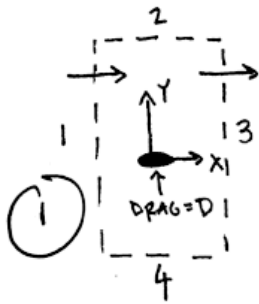
$U \int_{-\infty}^{\infty} \frac{\partial u_1}{\partial x} dy = \nu \int_{-\infty}^{\infty} \frac{\partial^2 u_1}{\partial y^2} dy$

$U \frac{d}{dx} \int_{-\infty}^{\infty} u_1 dy = \nu \left[\frac{\partial u_1}{\partial y} \right]_{-\infty}^{+\infty}$ B.C.: $u_1 \rightarrow 0$ AS $y \rightarrow \pm \infty$

(") = 0

$\Rightarrow \int_{-\infty}^{\infty} u_1 dy = \text{CONST.}$

e) (5 pts) Using control volume arguments, show how (1) relates to the drag on the body.



- STEADY FLOW, SO X-MOMENTUM IN THE VOLUME IS CONSTANT. ALSO PRESSURE = CONST.
- LET SIDES 2 AND 4 BE AT $\pm \infty = Y$, SO $\vec{u} = (U, 0)$ THERE.

- X-MOMENTUM FWX INTO VOLUME THROUGH SIDE 1 =
 DRAG + X-MOMENTUM FWX OUT OF VOLUME THROUGH SIDE 3

$$\Rightarrow \int_{-\infty}^{\infty} \rho U^2 dy = D + \int_{-\infty}^{\infty} \rho (U + u_1(x, y))^2 dy = D + \int_{-\infty}^{\infty} \rho [U^2 + 2Uu_1(x, y) + u_1(x, y)^2] dy$$

$$\Rightarrow \int \rho U^2 \text{ CANCELS, SO } D = -\rho \int_{-\infty}^{\infty} [2Uu_1 + u_1^2] dy \quad u_1 \ll U$$

$$= -\rho U^2 \int_{-\infty}^{\infty} \frac{u_1}{U} dy \Rightarrow \text{CONST.} = \int_{-\infty}^{\infty} u_1 dy = -\frac{D}{2\rho U}$$

f) (5 pts) Assume a similarity solution for the velocity of the form

$$u_1 = F(x)f(\eta), \text{ where } \eta = \frac{y}{g(x)}$$

For this assumption to be valid, how must g vary with x ?

$$\text{NEED: } \frac{\partial u_1}{\partial x} = F'(x)f(\eta) - F(x)\frac{g'(x)}{g(x)}\eta f'(\eta), \quad \frac{\partial^2 u_1}{\partial y^2} = \frac{F(x)}{g(x)^2} f''(\eta)$$

$$\text{B.L. EQ. FROM PART C) BECOMES } U(F'f - F\frac{g'}{g}\eta f') = \nu \frac{F}{g^2} f''$$

$$\text{MULTIPLY THROUGH BY } \frac{g}{Fg'} \Rightarrow U \frac{Fg'}{Fg'} f - U\eta f' = \nu \frac{1}{gg'} f''$$

$$\text{FROM TERM ON RHS, } g \cdot g' = \text{CONST.} \Rightarrow \int \frac{1}{2} g^2 = Cx + D$$

$$\Rightarrow \underline{g \sim x^{1/2}}$$

Bonus questions from the world of sports.

1. Which three drivers were the first to qualify at over 200 mph for the Indy 500?
 - a) Johnny Rutherford, Al Unser Sr., Rick Mears
 - b) Tom Sneva, Al Unser Sr., Johnny Rutherford
 - c) Bobby Unser, A.J. Foyt, Gordon Johncock
 - d) Rick Mears, Tom Sneva, Danny Ongais

2. Which of these alpine skiing stars represented Luxembourg?
 - a) Pirmin Zurbriggen
 - b) Hanni Wenzel
 - c) Franck Piccard
 - d) Marc Girardelli

3. The infield fly rule pertains in which of these situations (could be more than one)?
 - a) Bases loaded, two out
 - b) Runners on first and third, no outs
 - c) Runners on first and second, one out
 - d) Runners on first and second, two out

4. Wayne Gretzky's single-season NHL record for goals is:
 - a) 83
 - b) 86
 - c) 88
 - d) 92

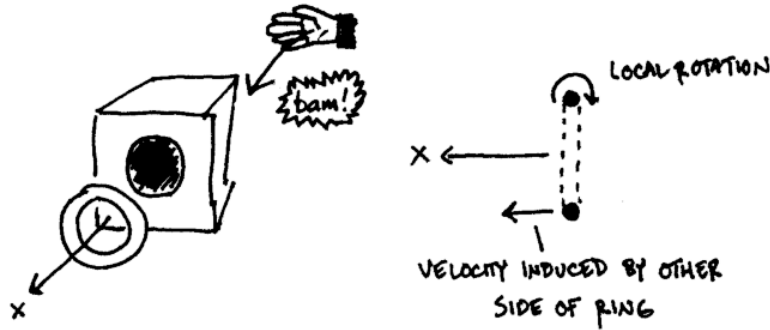
5. Which style of play was Martina Navratilova famous for?
 - a) Baseline
 - b) Serve and volley
 - c) Selective attacking
 - d) Mash the ball senseless, then hit the weights

ME 563 - Intermediate Fluid Dynamics - Su
 Final Exam - 13 May 2002
 120 points

Name _____

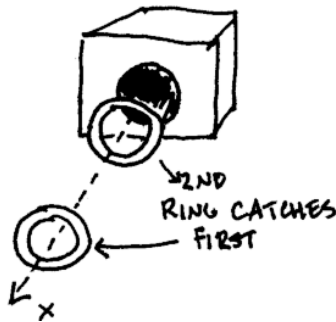
Problem 1 - Short answers. (Total 50 pts)

- (a) (25 pts) In class we saw a demonstration of vortex rings, generated when fluid was forced out of a box through a circular orifice. Vortex rings are able to propagate a long distance through the mechanism illustrated in the figure. We can approximate the vortex ring as a circular vortex



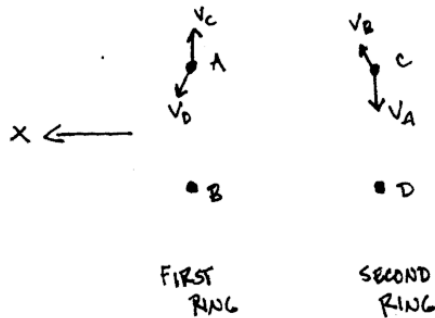
tube, and we know that in the inviscid case, vortex lines (and thus tubes) move at the velocity of the fluid. At any part of the ring, the rotation induces the portion of the ring on the opposite side of the circle to move in a direction perpendicular to the plane of the ring. In this way, the ring can propagate indefinitely without change in size or shape, in the absence of viscosity.

When two identical vortex rings are produced successively, the second vortex ring catches up with the first. Ignoring viscous effects, explain how this happens. You can do this without equations, but you will need to use Helmholtz's vortex theorems.



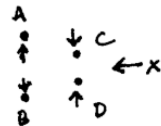
(Space for part (a).)

- THE EXPLANATION BEGINS BY CONSIDERING THE VELOCITY INDUCED BY EACH RING ON THE OTHER. CONSIDER:



(v_c IS THE VELOCITY INDUCED ON 'POINT' A BY POINT C; v_d IS THE VELOCITY INDUCED ON POINT A BY POINT D; ETC.)

- $|v_c| > |v_d|$ SINCE POINT C IS CLOSER TO A THAN D IS
 \Rightarrow A IS INDUCED TO MOVE ^{WITH A COMPONENT} IN THE X-DIRECTION AND A COMPONENT AWAY FROM THE X-AXIS, BY THE SECOND RING
- SIMILARLY, $|v_a| > |v_b|$: C IS INDUCED TO MOVE WITH A COMPONENT IN THE X-DIRECTION AND A COMPONENT TOWARD THE X-AXIS
- BY SYMMETRY, THE INDUCED MOTION IN THE X-DIRECTION IS THE SAME FOR BOTH RINGS. SO IN A REFERENCE FRAME MOVING WITH BOTH RINGS, THE FIRST RING WIDENS WHILE THE SECOND GETS SMALLER.



- SMALLER RINGS HAVE HIGHER SELF-INDUCED VELOCITY \rightarrow INDUCED VELOCITY GOES LIKE $\frac{\Gamma}{r}$, AND Γ IS CONSTANT FOR VORTEX TUBES
 $\Gamma \leftarrow$ CIRCUMFERENCE
 (BY HELMHOLTZ'S 2ND THEOREM) \Rightarrow SMALLER Γ MEANS FASTER RINGS

\Rightarrow THE 1ST RING SLOWS BY EXPANSION; THE 2ND ACCELERATES BY CONTRACTION, EVENTUALLY PASSING THE 1ST RING.

(b) (10 pts) Consider the transport equation for a conserved scalar quantity, ζ :

$$\frac{\partial \zeta}{\partial t} + \overbrace{\mathbf{u} \cdot \nabla \zeta}^{\text{inertial}} - \overbrace{D \nabla^2 \zeta}^{\text{diffusive}} = 0.$$

Let U be a characteristic velocity, L be a characteristic length scale, and Z be a characteristic scalar concentration. Derive the non-dimensional expression that describes the relative importance of the inertial and diffusive terms for the scalar transport equation (analogous to the Reynolds number for the Navier-Stokes equations).

WRITE $t^+ = \frac{t}{L/U}$, $\vec{u}^+ = \frac{\vec{u}}{U}$, $x_i^+ = \frac{x_i}{L}$, $\zeta^+ = \frac{\zeta}{Z}$

$\rightarrow \nabla^+ = L \nabla$

EQUATION BECOMES:

$$\frac{U}{L} Z \frac{\partial \zeta^+}{\partial t^+} + \frac{U}{L} Z \vec{u}^+ \cdot \nabla^+ \zeta^+ - D \frac{1}{L^2} Z \nabla^{+2} \zeta^+ = 0$$

MULTIPLY THROUGH $\frac{L}{UZ} \Rightarrow$

$$\frac{\partial \zeta^+}{\partial t^+} + \vec{u}^+ \cdot \nabla^+ \zeta^+ - \frac{D}{UL} \nabla^{+2} \zeta^+ = 0.$$

NON-DIMENSIONAL: IN NS, THIS WOULD BE $\frac{U}{\nu} = \frac{1}{Re}$

APPROPRIATE NON-DIMENSIONAL TERM IS $\frac{UL}{D}$.

(NOTE: THIS IS SOMETIMES WRITTEN

$$\left. \begin{array}{l} Pe = Re Sc = \frac{UL}{D} \\ \downarrow \quad \quad \downarrow \\ \text{Péclet \#} \quad \text{SCHMIDT \#,} = \frac{\nu}{D} \end{array} \right)$$

- (c) (15 pts) Consider an axisymmetric, turbulent jet flowing in the x -direction. The mean velocity in the x -direction is given by

$$u(x, r) = u_c(x)f(\eta), \text{ where } \eta = \frac{r}{\delta(x)},$$

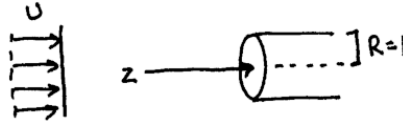
and, as we saw in class, $\delta = Cx$ and $u_c = D/x$, where C and D are constants. Now define $\dot{m}(x)$ as the mean mass flux across a surface of fixed x . How does \dot{m} depend on x ? Is the mass flux a constant? Whether it is or isn't, explain what's happening.

$$\begin{aligned} \dot{m}(x) &= \rho \int_0^{2\pi} d\theta \int_0^{\infty} dr \cdot r \cdot u(x, r) \\ &= 2\pi \rho \int_0^{\infty} dr \cdot r \cdot u_c f(\eta) \quad \delta \cdot d\eta = dr \\ &= 2\pi \rho u_c \int_0^{\infty} d\eta \cdot \eta f(\eta) \delta^2 \\ &= 2\pi \rho u_c \delta^2 \underbrace{\int_0^{\infty} d\eta \cdot \eta f(\eta)}_{\text{CONST.}} \end{aligned}$$

$$u_c \propto \frac{1}{x}, \quad \delta \propto x \Rightarrow \dot{m}(x) \propto x.$$

- THE MASS FLUX INCREASES LINEARLY WITH x , BECAUSE THE JET DRAGS FLUID ALONG WITH IT (THIS PROCESS IS CALLED 'ENTRAINMENT').

Problem 2. (Total 35 pts) Consider a viscous, incompressible fluid flowing with uniform velocity U in the z -direction. The fluid encounters a thin-walled, circular tube of radius $R = 1$ whose axis is aligned with the z -axis. We are interested in the steady flow that develops inside this tube.



Working in cylindrical coordinates, the appropriate Navier-Stokes equations are

$$(\mathbf{u} \cdot \nabla)u_r - \frac{u_\theta^2}{r} = -\frac{1}{\rho} \frac{\partial p}{\partial r} + \nu \left(\nabla^2 u_r - \frac{u_r}{r^2} - \frac{2}{r^2} \frac{\partial u_\theta}{\partial \theta} \right) \quad (1)$$

$$(\mathbf{u} \cdot \nabla)u_\theta + \frac{u_r u_\theta}{r} = -\frac{1}{\rho r} \frac{\partial p}{\partial \theta} + \nu \left(\nabla^2 u_\theta + \frac{2}{r^2} \frac{\partial u_r}{\partial \theta} - \frac{u_\theta}{r^2} \right) \quad (2)$$

$$(\mathbf{u} \cdot \nabla)u_z = -\frac{1}{\rho} \frac{\partial p}{\partial z} + \nu \nabla^2 u_z \quad (3)$$

$$\nabla \cdot \mathbf{u} = \frac{1}{r} \frac{\partial}{\partial r}(r u_r) + \frac{1}{r} \frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z} = 0 \quad (4)$$

making use of the following:

$$(\mathbf{u} \cdot \nabla) = u_r \frac{\partial}{\partial r} + \frac{u_\theta}{r} \frac{\partial}{\partial \theta} + u_z \frac{\partial}{\partial z}, \quad \text{and}$$

$$\nabla^2 = \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2}{\partial \theta^2} + \frac{\partial^2}{\partial z^2}.$$

- a) (8 pts) Since the boundary conditions are axisymmetric, it is reasonable to assume that the velocity field inside the tube will also be axisymmetric. Rewrite Eqs. (1), (2), (3) and (4) for an axisymmetric velocity field.

STEADY, NO TIME DEPENDENCE
 $\uparrow \qquad \qquad \qquad \uparrow$
 AXISYMMETRIC FLOW: $\vec{u} = (u_r(r, z, t), 0, u_z(r, z, t))$

$$\textcircled{1} \rightarrow u_r \frac{\partial u_r}{\partial r} + u_z \frac{\partial u_r}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial r} + \nu \left(\frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial u_r}{\partial r} \right) + \frac{\partial^2 u_r}{\partial z^2} - \frac{u_r}{r^2} \right)$$

$$\textcircled{2} \rightarrow 0 = -\frac{1}{\rho r} \frac{\partial p}{\partial \theta} \Rightarrow p \neq f_n(\theta), \text{ OBVIOUS SINCE AXISYMMETRIC}$$

$$\textcircled{3} \rightarrow u_r \frac{\partial u_z}{\partial r} + u_z \frac{\partial u_z}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial z} + \nu \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial u_z}{\partial r} \right) + \nu \frac{\partial^2 u_z}{\partial z^2}$$

$$\textcircled{4} \rightarrow \frac{1}{r} \frac{\partial}{\partial r} (r u_r) + \frac{\partial u_z}{\partial z} = 0$$

b) (15 pts) Immediately after the fluid enters the tube, a boundary layer will begin to develop on the tube's interior walls. In the boundary layer, let U be a characteristic value of u_z , and let u_z change by an amount of order U over a distance L in the z -direction. Also, let δ be a characteristic thickness of the boundary layer, where $\delta \ll L$. Working from the equations you developed in part a), derive the boundary layer equations for this flow.

- FIRST LOOK AT EQ. (4) FROM PART a). NOTE THAT ON THE WALLS OF THE TUBE, $r=1$, SO WE CAN WRITE (FOR THE PURPOSE OF ESTIMATING MAGNITUDES ONLY)

$$\approx \frac{\partial u_r}{\partial r} + \frac{\partial u_z}{\partial z} = 0 \quad \text{NEAR THE WALLS}$$

WANT TO KEEP BOTH TERMS: $\left| \frac{\partial u_z}{\partial z} \right| \approx \frac{U}{L}$

$$\Rightarrow \frac{\partial u_r}{\partial r} \approx \frac{|u_r|}{\delta} \approx \frac{U}{L} \Rightarrow |u_r| \approx \frac{U\delta}{L} \ll U \approx |u_z|.$$

\downarrow
 SINCE $u_r = 0$
 ON THE WALLS

\Rightarrow EQ. (1) NEGLIGIBLE COMPARED WITH (3).

- NOW LOOK AT (3).

LHS: $\left| u_r \frac{\partial u_z}{\partial r} \right| \approx \frac{U^2}{L}$, $\left| u_z \frac{\partial u_z}{\partial z} \right| \approx \frac{U^2}{L} \Rightarrow$ BOTH TERMS STAY.

RHS: COMPARE VISCOUS TERMS: $\left| \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial u_z}{\partial r} \right) \right| \approx \frac{U}{\delta^2}$ USING $r \approx 1$

$$\left| \frac{\partial^2 u_z}{\partial z^2} \right| \approx \frac{U}{L^2} \ll \frac{U}{\delta^2}, \quad \text{THIS TERM IS NEGLIGIBLE.}$$

$$\Rightarrow \left. \begin{aligned} u_r \frac{\partial u_z}{\partial r} + u_z \frac{\partial u_z}{\partial z} &= -\frac{1}{\rho} \frac{\partial p}{\partial z} + \nu \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial u_z}{\partial r} \right) \\ \frac{1}{r} \frac{\partial}{\partial r} (r u_r) + \frac{\partial u_z}{\partial z} &= 0 \end{aligned} \right\} \text{B.L. EQS.}$$

c) (12 pts) If we go far enough downstream in the tube, we expect that the flow inside will become parallel in the z -direction, and that u_z will have a parabolic profile. Estimate just how far we have to go in the tube for this so-called 'fully-developed flow' to pertain. Explain your reasoning.

- THE BOUNDARY LAYER GROWS AS THE FLOW MOVES DOWNSTREAM IN THE TUBE. THE FLOW WILL BE FULLY DEVELOPED WHEN THE LAYERS FROM OPPOSITE SIDES MERGE AND CEASE TO GROW. TO A FIRST APPROXIMATION, THIS WILL PERTAIN WHEN THE 'INDIVIDUAL' BOUNDARY LAYERS HAVE THICKNESS $\delta \approx R$, THE TUBE RADIUS.

- FROM PART (b), THE INERTIAL TERMS IN THE B.L. EQS. HAVE MAGNITUDE $\approx \frac{U^2}{L}$
THE VISCOUS TERM $\approx \nu \frac{U}{\delta^2}$] WE REQUIRE THESE TO BE COMPARABLE SO VISCOSITY IS IMPORTANT

$$\frac{U^2}{L} \approx \nu \frac{U}{\delta^2} \Rightarrow \delta \approx \left(\frac{\nu L}{U} \right)^{1/2}$$

FOR $\delta \approx R$, NEED $L \approx \frac{UR^2}{\nu}$ ← NEED TO GO ABOUT THIS FAR INTO THE TUBE FOR FULLY-DEVELOPED FLOW.

↓

$\left(\frac{\nu L}{U} \approx R^2 \right)$

Problem 3. (Total 35 pts) This problem concerns the steady, parallel flow in the x -direction of a viscous, incompressible fluid between plane boundaries at $y = 0$ and $y = h$. Assume that the flow is fully developed, so none of the flow properties depends on x .

a) (5 pts) Consider the internal energy of the fluid. Is there heat generated by the flow? If so, what happens to this heat, and if not, why not?

- HEAT IS GENERATED BY VISCOUS DISSIPATION
- SINCE THE FLOW TEMPERATURE DOESN'T INCREASE WITH x , AND THE FLOW IS STEADY (NO TIME DEPENDENCE), THE HEAT GENERATED MUST BE TRANSFERRED TO THE WALLS.

b) (5 pts) Suppose that the thermal conductivity of the fluid, k , is constant; also assume that the internal energy can be written $E = cT$, where c is the (constant) specific heat of the fluid and T is the temperature. Write the internal energy equation for the particular flow in this problem.

GENERAL INTERNAL ENERGY EQ. :

$$\rho \frac{\partial E}{\partial t} + \rho (\vec{u} \cdot \nabla) E = 2\mu e_{ij} e_{ij} + k \frac{\partial^2 T}{\partial x_j \partial x_j}$$

$$\rho u_j \frac{\partial E}{\partial x_j}$$

FOR OUR PROBLEM: $\frac{\partial E}{\partial t} = 0$, STEADY FLOW

$$\vec{u} = (u, 0, 0) \Rightarrow u_j \frac{\partial E}{\partial x_j} = u \frac{\partial E}{\partial x} = 0, \text{ FULLY DEVELOPED.}$$

$$e_{ij} = \begin{pmatrix} 0 & \frac{1}{2} \frac{\partial u}{\partial y} & 0 \\ \frac{1}{2} \frac{\partial u}{\partial y} & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}, \quad u = u(y) \text{ ONLY, PARALLEL FLOW.}$$

$$\Rightarrow \underline{\underline{0 = \mu \left(\frac{\partial u}{\partial y} \right)^2 + k \frac{\partial^2 T}{\partial y^2} \quad \left(\text{BY GEOMETRY, } \frac{\partial^2 T}{\partial x^2} = 0 \right)}}$$

- c) (15 pts) Show that there is no solution to the internal energy equation (from part b) for this problem in the case where the boundaries are thermally insulated, i.e. when $\partial T / \partial y = 0$ at $y = 0, h$.

CONSIDER $0 = \mu \left(\frac{\partial u}{\partial y} \right)^2 + k \frac{\partial^2 T}{\partial y^2}$.

- $\rightarrow \left(\frac{\partial u}{\partial y} \right)^2 \geq 0$ ALWAYS, SO $\frac{\partial^2 T}{\partial y^2} \leq 0$ ALWAYS.

- BUT $\frac{\partial T}{\partial y} = 0$ AT $y = 0$ AND h . $\frac{\partial^2 T}{\partial y^2} \leq 0 \Rightarrow \frac{\partial}{\partial y} \left(\frac{\partial T}{\partial y} \right) \leq 0$,

i.e. $\frac{\partial T}{\partial y}$ EITHER DECREASES OR STAYS THE SAME. SINCE

$$\frac{\partial T}{\partial y} (y=0) = \frac{\partial T}{\partial y} (y=h), \text{ THIS MEANS } \frac{\partial^2 T}{\partial y^2} = 0. \text{ BUT}$$

THEN, WE HAVE THE UNINTERESTING CASE OF $\frac{\partial u}{\partial y} = 0$ EVERYWHERE,

i.e. NO FLOW!

- SO THERE IS NO SOLUTION TO THE ENERGY EQUATION IN b) IF THE BOUNDARIES ARE THERMALLY INSULATED.

(• PHYSICAL EXPLANATION: HEAT IS GENERATED BY THE FLOW BY VISCOUS DISSIPATION, SO IF NO HEAT IS TRANSFERRED TO THE WALLS, EITHER

$\frac{\partial T}{\partial t}$ OR $\frac{\partial T}{\partial x}$ MUST BE NONZERO, WHICH VIOLATES THE ASSUMPTIONS.)

d) (10 pts) Suppose now that the boundary at $y = 0$ is stationary, the boundary at $y = h$ moves at a speed U in the x -direction, and instead of being thermally insulated, both boundaries are held at a temperature T_0 . What's the temperature field, $T(y)$, between the boundaries?

• N-S: $0 = \frac{\partial^2 u}{\partial y^2}$, $C = \frac{\partial u}{\partial y}$, $u = Cy + D$.

$$u = 0 \text{ at } y = 0, \quad u = U \text{ at } y = h \Rightarrow \underline{u = U \frac{y}{h}}.$$

$$\Rightarrow \frac{\partial u}{\partial y} = \frac{U}{h}$$

• ENERGY EQ.: $0 = \underbrace{\mu \left(\frac{U}{h}\right)^2}_{\text{CONST.}} + k \frac{\partial^2 T}{\partial y^2}$

$$-\frac{\mu}{k} \left(\frac{U}{h}\right)^2 = "A" = \frac{\partial^2 T}{\partial y^2}$$

$$\Rightarrow Ay + B = \frac{\partial T}{\partial y} , \quad \frac{1}{2}Ay^2 + By + E = T$$

$$T(y=0) = T_0 \Rightarrow E = T_0$$

$$T(y=h) = T_0 \Rightarrow \frac{1}{2}Ah^2 + Bh = 0 = -\frac{\mu}{2k}U^2 + Bh$$

$$\Rightarrow B = \frac{1}{h} \left(\frac{\mu U^2}{2k} \right)$$

$$\Rightarrow T(y) = -\frac{\mu}{2k} \left(\frac{U}{h}\right)^2 y^2 + \frac{1}{h} \left(\frac{\mu U^2}{2k} \right) y + T_0.$$

Bonus questions from film and television.

1. Before becoming a private detective, Pierce Brosnan's title character on "Remington Steele" was
 - a) A secret agent
 - b) A police officer
 - c) A classical musician
 - d)
2. Which school is Ashley Judd's alma mater?
 - a) University of Tennessee
 - b) Vanderbilt University
 - c) Caltech
 - d) *She's a well-known UK basketball fan*
3. Who first played Jack Ryan, the recurring hero in Tom Clancy's novels, in the movies?
 - a) Harrison Ford
 - b) Wesley Snipes
 - c) *In 'The Hunt for Red October'*
 - d) Kurt Russell
4. In the movie "Random Hearts", Kristin Scott-Thomas plays
 - a) A Senator
 - b)
 - c) A Cabinet secretary
 - d) A casino blackjack dealer
5. Which one of these actresses did *not* star on "The Facts of Life"?
 - a) Lisa Whelchel
 - b) Molly Ringwald *appeared in the first season only*
 - c)
 - d) Nancy McKeon

530.621: Graduate Fluid Mechanics I
Midterm 1, Fall 1997

Time: 2 hrs. Open books & notes
Please write and sketch in legible fashion.

1a. (5 points) The enstrophy per unit volume of a flow is defined as $|\boldsymbol{\omega}|^2/2 = \omega_i \omega_i / 2$, where $\boldsymbol{\omega}_i$ is the vorticity vector $\boldsymbol{\omega} = \boldsymbol{\nabla} \times \mathbf{u}$. Consider a divergence-free velocity field, $\boldsymbol{\nabla} \cdot \mathbf{u} = 0$. Using index notation, prove the following identity:

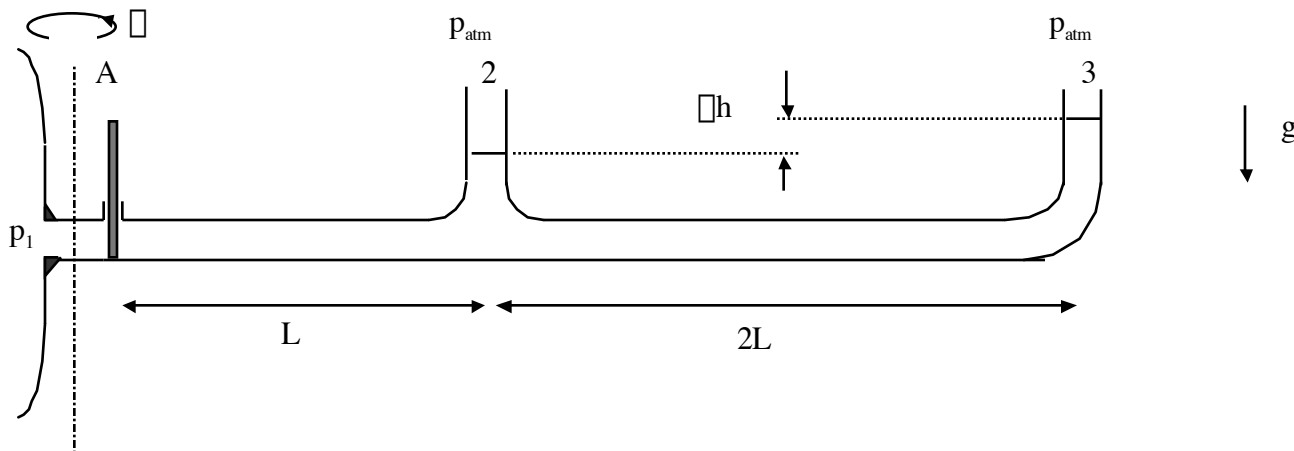
$$|\boldsymbol{\omega}|^2 = \boldsymbol{\nabla}^2 (|\mathbf{u}|^2/2) - \mathbf{u} \cdot \boldsymbol{\nabla}^2 \mathbf{u} + (\boldsymbol{\nabla} \boldsymbol{\nabla}) : (\mathbf{u} \mathbf{u}),$$

where $(\boldsymbol{\nabla} \boldsymbol{\nabla}) : (\mathbf{u} \mathbf{u}) = \partial_i \partial_j (u_i u_j)$, and $\partial_i = \partial/\partial x_i$.

1b. (5 points) Consider an incompressible, Newtonian, viscous fluid but with spatially varying viscosity $\mu(\mathbf{x})$. Prove, using index notation, the following statement: "The total viscous force \mathbf{f}_v obtained by integrating the differential force (generated by the viscous stress) over a closed surface S bounding a control volume V, is given by:

$$\mathbf{f}_v = \int_V \boldsymbol{\nabla} \cdot (\boldsymbol{\tau} \boldsymbol{\nabla}^2 \mathbf{u}) dV + \int_V \boldsymbol{\nabla} \mu \cdot (\boldsymbol{\nabla} \mathbf{u} + \boldsymbol{\nabla} \mathbf{u}^T) dV."$$

2. (20 points) A sprinkler system with two vertical exit pipes is externally driven. The pressure at the inlet is p_1 (known), and the angular velocity of the system is constant, equal to Ω (also known). All pipe diameters are constant, equal to D . Lengths of the two segments are L and $2L$, as shown in figure below. Incompressible fluid, with density ρ .



Distances between tank entrance, axis of rotation and gate, and the vertical heights at 2 and 3, are all very small compared to L . (cont'd on next page).

(a) Initially, at $t < 0$, the gate valve at A is closed and there is no flow. Determine the height difference between fluid levels at exits 2 and 3.

(b) At $t = 0$, the gate is suddenly opened completely. For any $t > 0$, formulate appropriate equations that will allow you to obtain the velocity u_2 and u_3 at exits 2 and 3. Assume uniform, inviscid flow inside the pipes, and neglect the length of the vertical portions as compared to the length of the horizontal portions. Also, for this and next parts (b & c) neglect gravity.

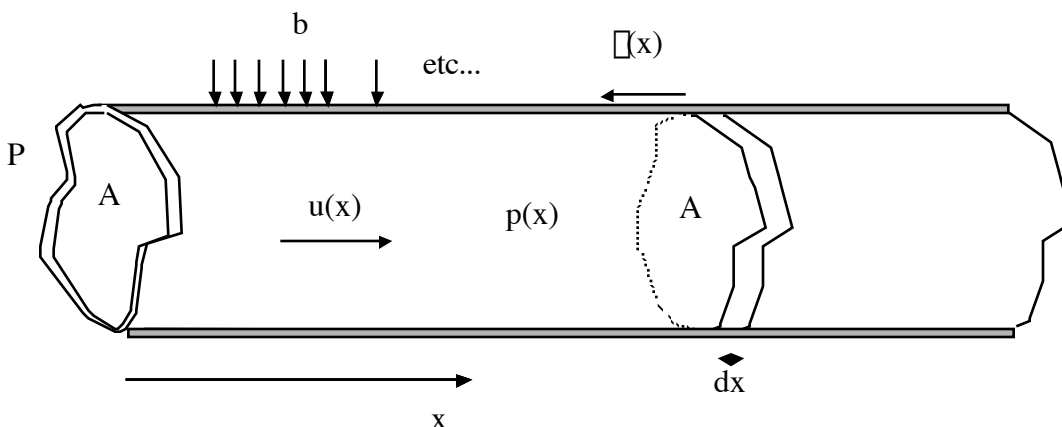
(c) Find expressions from which to determine the horizontal (parallel to the pipe) and vertical reaction forces at any time $t > 0$, acting at the weld near A. Neglect moments of inertia of the pipe material, but not of the fluid.

3. (10 points) Consider a porous pipe of constant cross-sectional area A and perimeter P . The flow rate into the pipe across the pipe wall is a constant b (per unit area of pipe surface, units of $(\text{m}^3/\text{s})/\text{m}^2$). It does not depend on x . The flow enters the pipe normal to the streamwise direction. Assume the wall shear stress is a known function of x , namely $\tau(x)$. We wish to find the streamwise velocity distribution $u(x)$ and pressure gradient $\partial p/\partial x(x)$. The flow is not inviscid, is steady, has constant density ρ , and inside the pipe the flow can be assumed to be unidirectional in the streamwise x -direction.

Use a thin control volume as shown in the figure (entire cross-section by dx) to:

(a) (5 points) Derive a differential equation relating $u(x)$ to x , A , P , b . Solve, using $u(x=0) = U$.

(b) (5 points) Derive another differential equation relating the pressure gradient to $u(x)$, the known $\tau(x)$, and the other parameters of the problem. (You need not solve this differential equation).



Computer Assignment 2 Step-by-Step Instructions

1 Introduction

As in the first project, **GAMBIT** will be used to generate the mesh and **FLUENT** will be used to solve the corner flow. You may use **GAMBIT & FLUENT** on either the Windows PCs in the Kreiger computer lab or on `apserve1.hcf.jhu.edu` (Unix, for remote use). Instructions for both systems are given here, although unless you are comfortable with Unix, it is recommended that you follow the Windows instructions.

1.1(a) Starting GAMBIT in Windows

To start **GAMBIT**:

Start > Programs Engineering Applications > Fluent & Gambit > Gambit 2.0

On the main menu bar of **GAMBIT**:

File > New...

Enter the id of your project (these instructions assume you enter corner) and title. Click **Accept** then **Yes**.

All files will be created and saved in the directory `C:/SaveFilesHere`. The files can be moved using Windows Explorer later. To open Windows Explorer in Kreiger lab:

Start > Accessories & Media > Command Prompt
Type `explorer` at the prompt.

1.1(b) Starting GAMBIT in Unix

From your local system, open (secure) a connection to `apserve1` (you must already have an account). Make sure that your system will display graphics windows remotely requested by `apserve1`. If you use `ssh` to login to `apserve1`, the graphics will probably be automatically forwarded to your local machine.

At the shell prompt, type (nozzle will be the id of the project):

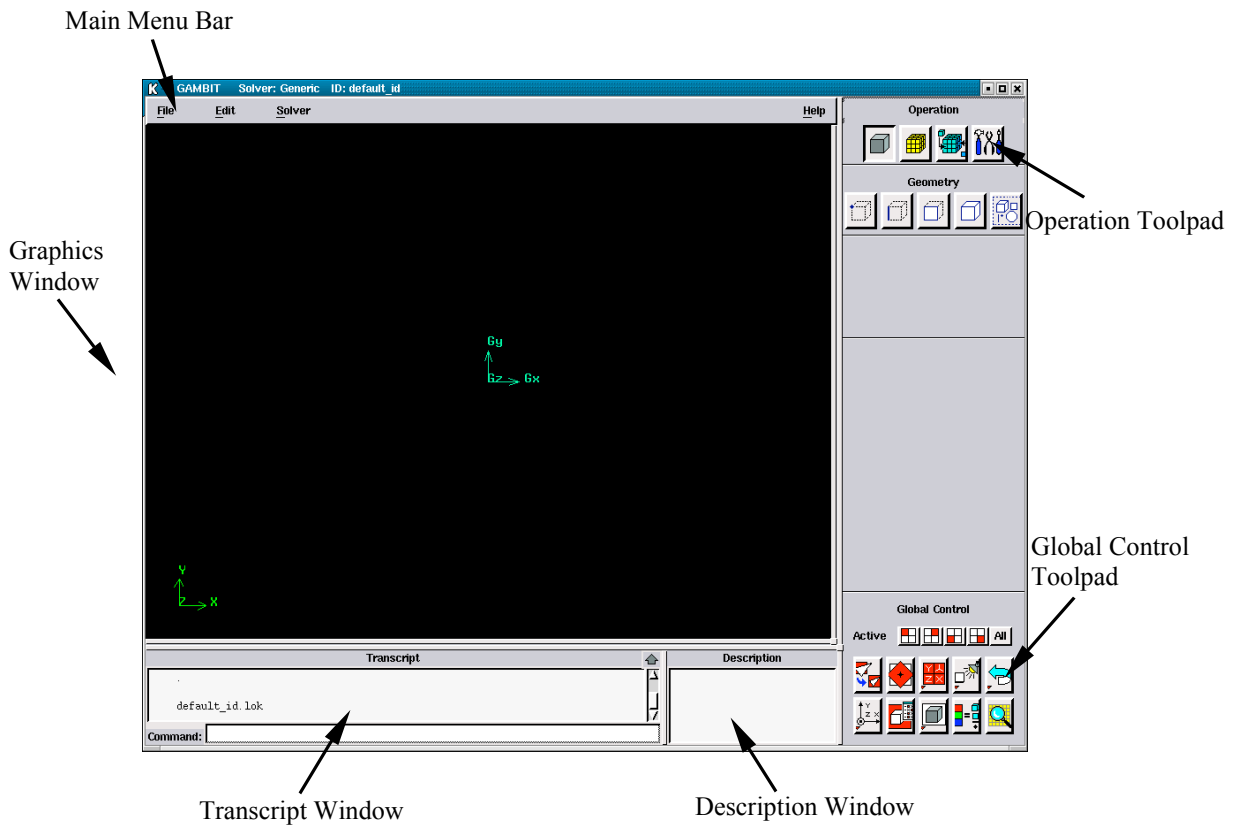
```
gambit nozzle &
```

The `&` symbol runs **GAMBIT** in the background and gives you back control of your interactive shell.

1.2 GUI Layout

The Graphical User Interface layout for **GAMBIT** is shown below. Sometimes the windows in this field are too long and the bottom part might be hidden under the taskbar at the bottom of the screen (or in Unix, it may be drawn off the bottom edge of the screen)--this will hide important buttons

such as **Apply** and **Create** . You can move the window up by clicking on the window name bar (in dark blue on Windows PCs in Kreiger) and dragging upwards.



1.3 Help/Undo

While using GAMBIT, any steps can be undone with the undo button found in the Global control toolpad.



The delete button is available most of the time under most options in the operation “Mesh command”. Use this as an alternative to the UNDO button to remove unwanted objects.





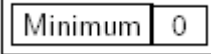
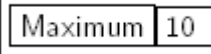
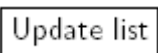


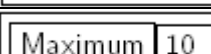






2 Grid Generation Using GAMBIT

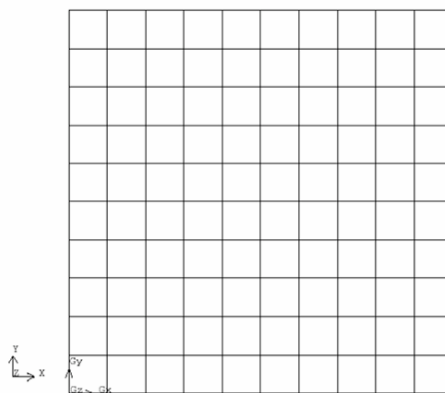
The first step in solving this fluid flow problem is to generate a grid that approximates the true shape of the corner. We begin by setting up a coordinate system, and then defining important vertices for the corner shape. These vertices are then used to define edges, which in turn are used to define surfaces.

2.1 Domain Borders

Begin by setting up the coordinate system and defining the borders of the flow domain.

Select the TOOLS COMMAND BUTTON	
Select the COORDINATE SYSTEM COMMAND BUTTON .	
Select the DISPLAY GRID BUTTON .	
Under Axis , make sure the diamond next to X is filled in.	
Set the number in the Minimum box to 0.	
Set the number in the Maximum box to 10.	
Update the list. (The name of this button might not show completely, it is directly to the right of the Maximum field)	
Under Axis , select the diamond next to Y .	
Set the number in the Minimum box to 0.	
Set the number in the Maximum box to 10.	
Update the list.	
Apply the changes (may need to move Display Grid window up to see the button).	
Close the Display Grid window	
Near the bottom right corner of the screen, in the Global Control Toolpad, click FIT TO WINDOW .	

The coordinate system is complete. Now your graphics window should look something like this:



2.2 Flow Geometry

Now we specify the corner geometry.

Select the **GEOMETRY COMMAND BUTTON**.



Select the **VERTEX COMMAND BUTTON**.



The **Create Real Vertex** window should appear. Under **Global**, make sure the $x,y,$ and z entries are all set to 0. Then press

This defines the point $(0,0,0)$ as a vertex of the nozzle and a **+** should appear at the point showing that it is a vertex. Now we define other vertices:

Under **Global**, set $(x,y,z)=(10, 0, 0)$ and press

Repeat with $(x,y,z)=(10, 10, 0)$ and press

Repeat with $(x,y,z)=(6.67, 10, 0)$ and press

Repeat with $(x,y,z)=(6.67, 5, 0)$ and press

Repeat with $(x,y,z)=(0, 5, 0)$ and press

2.3 Edge Generation

Select the **EDGE COMMAND BUTTON**.

The **Create Straight Edge** window should appear.

This is for defining edges using the vertices that were already created.

Holding the shift key down, click point $(0, 0)$ in the graphics window and, while keeping the shift key down, click the point $(10, 0)$ and then press



The line between the two points you just clicked should become highlighted. This process of holding shift and clicking on a point will be referred to as shift-click from now on.

Shift-click points $(10, 0)$ and $(10, 10)$ and press

Shift-click points $(10, 10)$ and $(6.67, 10)$ and press

Shift-click points $(6.67, 10)$ and $(6.67, 5)$ and press




Shift-click points $(6.67, 5)$ and $(0, 5)$ and press

Shift-click points $(0, 5)$ and $(0, 0)$ and press

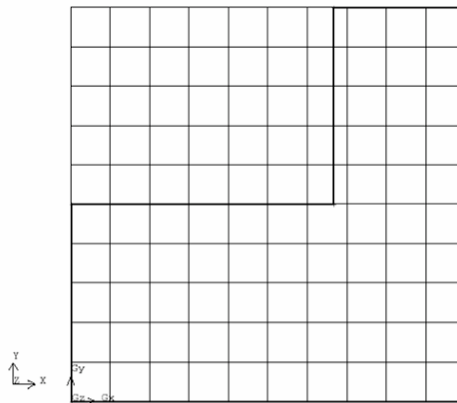
The Edge generation is complete.

2.4 Surface Definition

Now, the edges must be combined to define a surface (that will become the corner interior):





In the upper right, click the FACE COMMAND BUTTON .	
Click the arrow next to the Edges field, and move all to the picked column (use All->) and close the edge selection window	
Press	

The edge outlines should become highlighted and you should be able to see something like this in the graphics window:

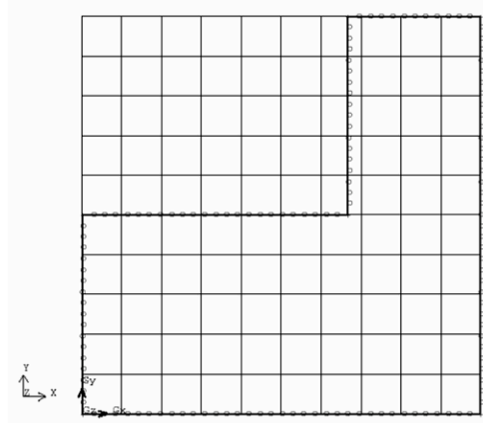


2.5 Mesh Creation

Now that we have the corner interior as a surface, we must “mesh” it. Meshing defines the discrete points at which our flow variables will be defined. We start by meshing the edges of the surface:

Near the top right, click the MESH COMMAND BUTTON (yellow color).	
Click the EDGE COMMAND BUTTON .	
Click the arrow to the right of the Edges field, move edge.1 to the picked column and close the Edge List window	
Under Spacing , change Interval size to Interval count , enter N_x given to you in the problem statement sheet, and press	
Repeat, but with edge.2 and an interval count of N_y	Don't forget for each edge to press
Repeat, but with edge.3 and an interval count of $N_x/3$	
Repeat, but with edge.4 and an interval count of $N_y/2$	
Repeat, but with edge.5 and an interval count of $2N_x/3$	
Repeat, but with edge.6 and an interval count of $N_y/2$	

Now you should see something like this:



Now click the **FACE COMMAND BUTTON**.



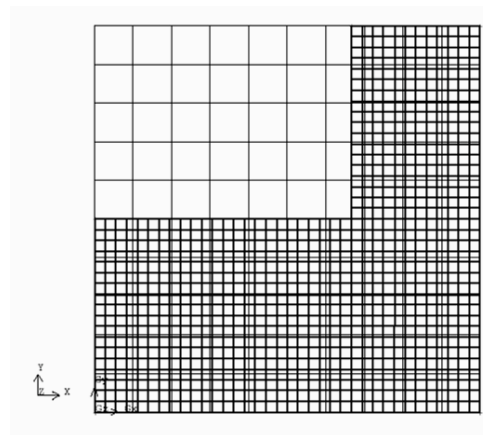
The **Mesh Faces** window will appear. Under **Faces**, pick face.1 then close the face selection window

Close

Apply the mesh face

Apply

We have now constructed a computational mesh for our domain. You should see something like this:



2.6 Edge Specification

We must now specify the type of each edge (inlet, wall, outlet, etc.)

From the main menu bar click

Solver > FLUENT 5/6

Now click the **ZONES COMMAND BUTTON** (blue color)



Click the **SPECIFY BOUNDARY TYPES COMMAND BUTTON** (white color).



Next to **Entity:**, change Faces to Edges.

Click on the arrow next to the Edges button and pick edge.6.

Change **Type:** to VELOCITY_INLET and press **Apply**. If the VELOCITY_INLET option is hidden under the taskbar, select SYMMETRY instead but do not Apply, and then click **Type** again, the menu will appear higher than before and you can select VELOCITY_INLET.

Apply

Repeat, but with edges 1 and 2 (move them both to the picked column) and with type WALL and press

Apply

Repeat, but with edge.3 and type OUTFLOW and press

Apply

Repeat, but with edges 4 and 5 and type WALL and press

Apply

The mesh is now ready.

2.7 Mesh Export

You now have to save the mesh and export it:

From the main menu bar click

File > Save

From the main menu bar click

File > Export > Mesh...

IMPORTANT: make sure to press the button **Export 2d Mesh**.

The default name of the mesh file will be corner.msh, you can keep it and then press **Accept**.

From the main menu bar click

File > Exit

If you do not exit **GAMBIT** properly you might have problems restarting the program for a short time (due to licensing issues).

3 Corner Flow Computation with FLUENT

Now that the grid has been generated, we use **FLUENT** to specify the velocity at the inlet and then calculate flow on this grid. The flow is assumed to be fully developed channel flow at the inlet, with a specified flow rate Q . To define the inlet profile, a short program written in C++, must be used.

3.1 Inlet Profile Program Listing

The program listing is given below, and the first step is to copy this into a file (call it something like "profile-u.cpp"). It is **NOT** necessary to compile the program, since **FLUENT** can interpret this file.

NOTE: YOU WILL HAVE TO CHANGE THE FLOW RATE Q IN THE FILE TO THE ONE GIVEN TO YOU IN THE ASSIGNMENT SHEET.

If you have problems email me at yili@pegasus.me.jhu.edu and I can email you the file.

Make sure the file is in your working directory.

```
#include "udf.h"
DEFINE_PROFILE(inlet_x_velocity, /* function name */
               thread, /* thread */
               nv) /* variable number */
{
    real Q; /* flow rate (m^3/s) */
    real height; /* height of channel inlet (m) */
    real depth; /* depth of channel (m) */
    face_t f;
    real x[ND_ND];
    Q=1.e-5; /* enter your flow rate here */
    height=5.e-5;
    depth=0.01;
    /* loop over each of the faces of this zone */
    begin_f_loop (f,thread)
    {
        F_CENTROID(x,f,thread); /* make sure you get the ()'s right here!! */
        F_PROFILE(f,thread,nv) = 6.*Q/(pow(height,3)*depth)*x[1]*(height-x[1]);
    }
    end_f_loop (f,thread)
}
```

3.2(a) Start FLUENT in Windows

To start FLUENT:

Start > Programs > Engineering Applications > Fluent & Gambit > FLUENT6.0

The **FLUENT Version** should appear. Select 2d and press **Run**.

3.2(b) Start FLUENT in Unix

At the shell prompt, type:

```
fluent &
```

A new window should appear. Enter 2d next to version>

3.3 Flow Computation

On the menu bar click:

```
File > Read > Case...
```

This will open the **Case** submenu, select corner.msh (or whatever you called your mesh) under **Files** and press **OK**.

On the menu bar click:

Grid > Scale...

This opens the Scale submenu. Change the scale factors to

x: 1e-5
y: 1e-5

then press **Scale** (make sure you only press this button once).

This corresponds to a scale of 100 μm , which is the dimension of our micromachine.

Now **Close**.

We must now define the material; we are using air at 20°C which is the default material.

On the menu bar click:

Define > Materials...

Set the properties to 20°C, i.e. Density = 1.205 kg/m³ and Viscosity = 1.81e-5 kg/m-s

Close the **Materials** window **Close**.

On the menu bar click:

Define > User-Defined > Functions > Interpreted...

The **Interpreted UDFs** window should appear.

Type in “profile-u.cpp” (or whatever you called the .cpp file) under **Source File Name**.

Press **Compile**

In the main **FLUENT** window you should see a long line starting with cpp and ending profile-u.cpp, make sure the line does not have any error messages. An error message indicates that there is probably a typo in the .cpp file.

Close the **Interpreted UDFs** window **Close**

On the menu bar click:

Define > Boundary Conditions

Select the velocity_inlet entry then press the **Set...**

Make sure the **Velocity Specification Method** is **Magnitude, Normal to Boundary**.

Leave the value in the **Velocity Magnitude** window at 0.

In the field to the right of **Velocity Magnitude** change it from **Constant** to **udf inlet_x_velocity**.

Press **OK**

Close the **Boundary Conditions** window **Close**.

On the menu bar click: Solve > Initialize > Initialize...

Press **Init**

Close

On the menu bar click: Solve > Iterate...

Set the number of iterations to 20 and press **Iterate**

This will start the solver and execute 20 iterations. Some results will be displayed in the main FLUENT window.

Keep iterating in steps of 20 (just re-press the **Iterate** button) until the main FLUENT window displays the message **solution is converged**.

Close the iterate window.

Now the computational part is done.

3.3 Post-Processing

Post-processing must be performed to extract the desired data. First we have to change the function of the mouse buttons.

On the menu bar click: Display > Mouse buttons...

Change the function of the right mouse button to mouse-zoom.

Click **OK**

3.3.1 Grid Display

Now we plot the grid

On the menu bar click: Display > Grid...

Then press **Display**.

A window containing the grid should appear. Zoom in by windowing the grid while holding down the right mouse button. Start the window at the upper left and end at lower right.

If you need to zoom out, hold the right mouse button and draw a window going from lower left to upper right.

3.3.1.1 Save the Plot

On the main FLUENT window menu bar click: File > Hardcopy ...

Select the JPEG format (or PostScript, etc.) for the plot.

Select **Gray scale** (if you want to print on a grayscale printer) or color for the **coloring** option.

Leave all other options as is and click **Save...**.

Enter the name you want to call your plot and press **OK**. Make sure you add the extension jpg to your filename. If you don't you can edit the filename later in Windows Explorer or just treat the file as a jpg.

The figure will have the same size as the plot on the screen, you can modify the size of your plot window before saving the hardcopy.

3.3.2 Velocity Vectors

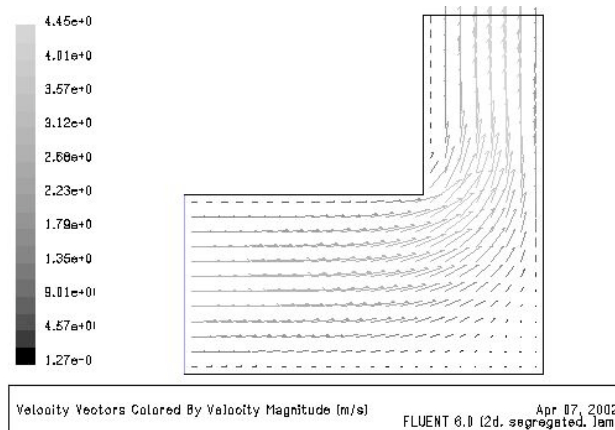
On the menu bar click: Display > Vectors...

The **Velocity...** and **Velocity Magnitude** options will be selected by default; keep them.

You might need to change the scale to get a nice looking plot, start by testing a **Scale** of 1.

Press **Display**

Once you get a nice plot, save the plot as you did in section 3.3.1.1. The velocity vector plot should look like this:



3.3.3 Streamlines

While keeping the velocity vector plot open:

On the menu bar click: Display > Contours...

Change the **Contours of** to the **Velocity...** option.

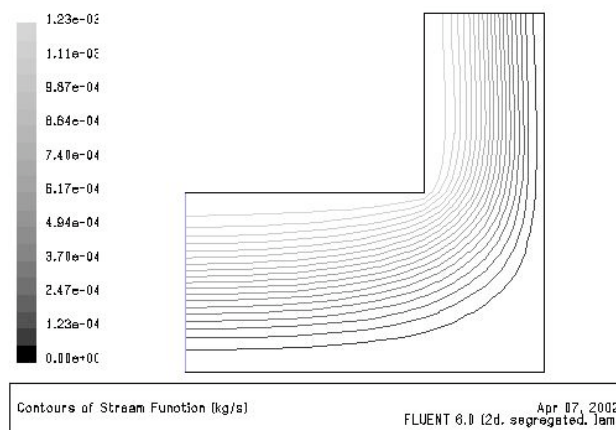
Change the second box to Stream Function option.

Set **Levels** to 30.

Click **Display**.

Save the plot as you did in section 3.3.1.1.

The plot should look like:



3.3.4 X-Velocity Profiles

To output x-velocity profiles, we first define a few stations (x-location) for the profiles:

On the menu bar click: Surface > Isosurface...

In the **Surface of Constant** box, select **Grid...** in the upper box.

In the second box select **X-Coordinate**.

In the **Iso-Values** box, enter the x coordinate where you want a velocity profile and then press **Create**.

For this problem it would be good to create stations at:

$X=2e-5$; $X=4e-5$; $X=6e-5$; $X=8e-5$.

And with **Y-Coordinate** selected in the second box:

$Y=2e-5$; $Y=4e-5$; $Y=6e-5$; $Y=8e-5$.

Note the names of the lines that are being created in the **New Surface Name** box. Press **Create** each time you change the value in the **Iso-Values** box.

Close the Iso-surface window.

We now want to display the velocity profiles along the lines we just defined.

On the menu bar click: Display > Contours...

In **Contours of** select **Velocity...**

In the second box select **X Velocity**.

Check the box next to **Draw Profiles** .
The **Profile Options** window will pop up.

Select a scale factor of $1e-6$ (You may need to play with the scale factor to get a nice plot, depending on Q and the number of vertical lines you created).

After closing the draw profiles window, it can be opened again by de-selecting and re-selecting the **Draw Profiles** option.

Press **Apply**.

And then **Close** the profile option window.

Under surfaces, select velocity inlet and ALL of your **vertical** lines (with names like x-coordinate-5, etc.).

Press **Display**.

Save the plot as you did in section 3.3.1.1.

To generate the profiles of the y-velocity, repeat, but:

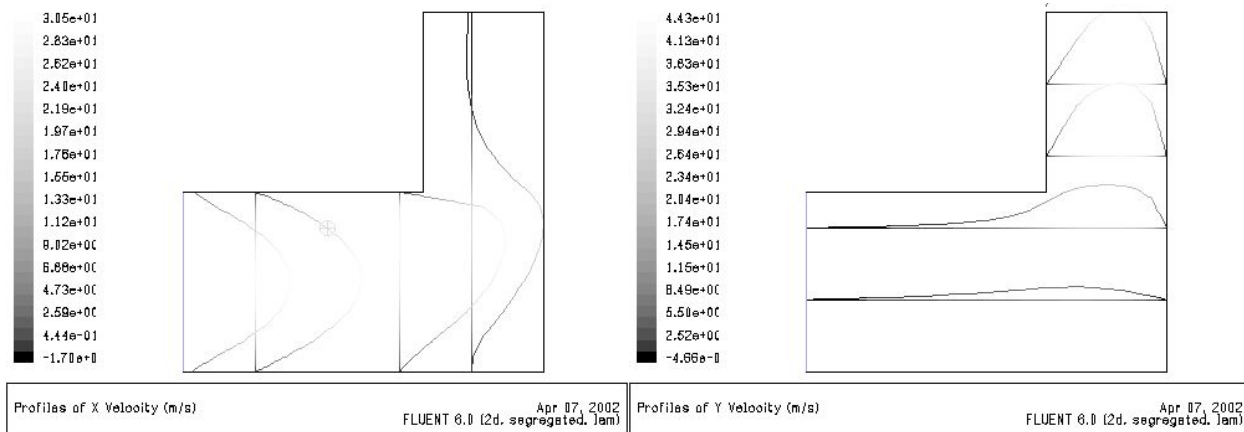
(i) Change **X-Velocity** to **Y-Velocity**.

(ii) Change selected surfaces to: OUTFLOW, and ALL of your **horizontal** lines (with names like y-coordinate-5, etc).

(iii) In the **Profile Options** window where you select the **Scale**, change the **Scale** to $5e-7$ and change **projection direction** to (0,1,0) (previously it was (1,0,0)).

Press **Display**.

The plots should look like this:



x-velocity profiles

y-velocity profiles

3.3.5 Forces on Wall

One way to find the force on the wall is to begin by extracting the velocity near the wall, and use this to estimate the derivative of the velocity profile at the wall. This information can then be used to calculate the wall shear stress and hence the total force on the wall. FLUENT does have the capability to calculate derivatives and forces automatically, and you may want to use these as a rough check of your results.

The first steps are to generate iso-surfaces of constant x-coordinate and constant y-coordinate near the wall. We will extract the velocity along these lines. Follow the steps for creating these iso-surfaces given above, and create the constant y-coordinate line at $y = 50 - (\Delta y/2)$ m, where Δy is the vertical spacing of your mesh. Likewise, create the constant x-coordinate line at $x = 66.67 + (\Delta x/2)$ m, where Δx is the horizontal spacing of your mesh. Note that $\Delta x = L_x / (N_x - 1)$, and similarly for Δy .

3.3.5.1 Horizontal Line

We'll start with the line along the horizontal part of the wall:

On the menu bar click: Plot > XY Plot...

The **Solution XY Plot** window will pop-up.

In the upper-right corner under **Y Axis Function** select **Velocity...** in the first box, and **X-velocity** in the second box.

Under **Surfaces**, select only the line you created at $y = 50 - (\Delta y/2)$ m and press **Plot**.

Make sure the values look reasonable and make a note of the units (since these won't be in your text file). Note that in calculating the derivatives and forces, you won't be using the points past where the wall turns, i.e. in this case we won't be using $x > 66.7$ μm .

Keep the plot open. We will now save the data in a text file.

3.3.5.2 Write to File

In the **Solution XY Plot** window check the box next to the option write to file.

Click the **Write...** button that will appear at the bottom of the window.

Call the file something like horizontal-line.xy and press **OK**.

You can view this file with any text editor or import it into a spreadsheet program (e.g. Excel).

3.3.5.3 Vertical Line

Now for the line near the vertical wall.

On the menu bar click: Plot > XY Plot...

The **Solution XY Plot** window will pop-up. De-select the writing option: write to file

Under Plot Direction change (X,Y)=(1,0) to (0,1).

In the upper-right corner under **Y Axis Function** select **Velocity...** then **Y-velocity**.

Under Surfaces, select only the line you created at $x = 66.67 + (\Delta x/2)$ m and press **Plot**.

Make sure the values look reasonable and make a note of the units (since these won't be in your text file). Keep in mind you'll only use the values that are actually near a wall (as above).

3.4 Finalizing

Now we are done, we just have to save our work:

On the menu bar click: File > Write > Case & Data...

The default name corner.cas should be fine.

Press **OK**

On the menu bar click: File > Exit

3.5 Important Notes

It would be best if you take all of the files you created in C:/SaveFilesHere (on Windows only), and either save them on a floppy diskette or ftp them somewhere else and then erase them from the PC. At least move them into a new folder that you create under C:/SaveFilesHere.

If you have money on your JHUID card, you can print in BW or color in the Krieger lab. BW printouts are 5c/page, color printouts are 25c/page.

Good luck!

ME 563 - Intermediate Fluid Dynamics - Su

Some review materials for the first exam 3/1/02

Note: Review session Wednesday 2/27, 5:30–7 pm, 318 ME (normal class room).

1 Framework of ideas

With all of the equations and theorems we've covered, it is important that we be able to organize the various ideas into a clear framework, so that we know what assumptions we need to make to apply which results, and so we know under what circumstances we can apply which assumptions. The property that is common to all flows we have discussed in this class is that they are **incompressible**. The major distinction in the class so far is between **inviscid** and **viscous** fluids. To recap:

Ideal Fluids

- The only force acting across a surface element in the fluid is the pressure
 - Flows are driven by pressure gradients ($\nabla p \neq 0$), or gravity
 - Incompressible: $\nabla \cdot \mathbf{u} = 0$
- **Euler's equations**
 - Kelvin circulation theorem, Helmholtz vortex theorems, Cauchy-Lagrange theorem
 - **Steady flow:** Bernoulli streamline theorem
 - If also **irrotational:** Bernoulli theorem for entire flow
 - **2-D flow:** vorticity conserved for fluid elements
 - If also **steady:** vorticity constant on streamlines

Viscous, incompressible fluids

- Besides pressure, there are also viscous forces acting across surface elements in the fluid
 - Flows observe the **no-slip condition** at solid boundaries
- **Navier-Stokes equations**
 - Viscous forces: **shear** forces (resist 'sliding'), **normal** forces (resist compression/expansion)
 - Viscous forces = additional *flow drivers*
 - Because of no-slip condition, viscous effects most strongly felt near walls - viscous-affected region = *boundary layer*
 - Viscosity diffuses momentum and vorticity - analogous to diffusion of heat

As we have discussed, we are generally free to use ideal fluid concepts until *boundary layer* effects become significant. Obviously, this can happen if there is a solid boundary in the problem.

In the wing example, Acheson points out that ideal fluid theory is pretty good at predicting the lift for small angles of attack, α , but that ideal fluid theory fails when the angle of attack gets large. Even though real fluids (e.g. air) are viscous, at small α ideal fluid theory works OK because the boundary layer, where viscous effects are important, is thin and occupies a very small portion of the flow. At higher α , the wing experiences boundary layer separation, so the boundary layer occupies a much larger portion of the flow and viscous effects can no longer be neglected.

Another instructive example is the one in the book about the impulsively moved plane boundary (§2.3). In that case, the boundary and fluid above it are initially at rest. Suddenly, the boundary starts moving at a fixed speed, which it then maintains for all time. As time passes, fluid that lies further and further away from the boundary gets accelerated, as viscosity allows the momentum (and vorticity) which is introduced to the fluid at the boundary (by the no-slip condition) to get diffused upward into the regions of the fluid that are essentially still at rest.

2 Similarity solutions

The impulsively moved plane boundary problem also introduced the idea of *similarity solutions*. In their most basic form, similarity solutions allow us to reduce the number of independent variables in a problem from two to one, by introducing a similarity variable that relates the original two independent variables. Then, the original partial differential equation, which involves two variables, can now be written as an ordinary differential equation, which involves just one variable. The value of the similarity solution approach arises because ordinary differential equations are much easier to solve, analytically or numerically.

Let's review what we did for the impulsively moved plane boundary, emphasizing the similarity aspects. Our original equation was

$$\frac{\partial u}{\partial t} = \nu \frac{\partial^2 u}{\partial y^2}, \quad (1)$$

which is a partial differential equation with independent variables t and y . In the notes, we argued for a similarity solution on physical grounds. The mathematical argument goes as follows. The variable y enters (1) as a second-order term, so if we define a new variable $y' = \alpha y$, where α is some constant, then (1) becomes

$$\frac{\partial u}{\partial t} = \nu \alpha^2 \frac{\partial^2 u}{\partial u'^2}. \quad (2)$$

The variable t enters as a first-order term, so if we define a new variable $t' = \beta t$, where β is some constant, (2) becomes

$$\beta \frac{\partial u}{\partial t'} = \nu \alpha^2 \frac{\partial^2 u}{\partial u'^2}. \quad (3)$$

By inspection, if $\beta = \alpha^2$, then (3) is the same equation as (1), except that the independent variables are t' and y' . This means that if we do the transformation

$$\begin{aligned} y' &= \alpha y \\ t' &= \alpha^2 t \end{aligned} \quad (4)$$

the mathematics of the problem defined by 1 is the same. We can also interpret the transformation (4) as meaning that the problem is unchanged if y/\sqrt{t} is unchanged, i.e. if $y'/\sqrt{t'} = y/\sqrt{t}$. This then

suggests that the solution of the equation (1) can be expressed in terms of the variable $\eta = y/\sqrt{t}$. We are free to multiply this resulting variable by any constant we like (except zero), since that won't affect the dependence of the problem on y and t . We can choose, as the book does, to let $\eta = y/\sqrt{\nu t}$, because this removes the constant ν from (1). This is also convenient because η defined this way is dimensionless.

With this new independent variable, we then look for a solution of the form

$$u = Uf(\eta)$$

or, as the book does, $u = f(\eta)$. There is no difference in these two approaches, since U is a constant in the problem. With either form for u , (1) becomes

$$f'' + \frac{\eta}{2}f' = 0, \tag{5}$$

which is an ordinary differential equation involving the single independent variable, η . As we saw in class, there's no problem solving (5), subject to the proper boundary conditions.

This process may seem kind of arbitrary, since it seems like we're guessing the form of the solution. What's to stop us from guessing some other solution? To which the response is twofold. First, if you guess a solution and it's right, then you're done, because you can't have different solutions to the same problem if the problem is fully specified¹. Secondly, let's look what happens if we guess a different similarity solution for (1). Say, let's look for a solution of the form

$$u = f(\eta) \quad \text{where} \quad \eta = \frac{y}{t}. \tag{6}$$

Then we have

$$\begin{aligned} \frac{\partial u}{\partial t} &= \frac{df}{d\eta} \frac{\partial \eta}{\partial t} = -f'(\eta) \frac{y}{t^2} \\ \frac{\partial u}{\partial y} &= \frac{df}{d\eta} \frac{\partial \eta}{\partial y} = f'(\eta) \frac{1}{t} \\ \frac{\partial^2 u}{\partial y^2} &= f''(\eta) \frac{1}{t^2}. \end{aligned}$$

Inserting these into (1), we get

$$\begin{aligned} -f' \frac{y}{t^2} &= \nu f'' \frac{1}{t^2}, \quad \text{which gives} \\ f'' + \frac{y}{\nu} f' &= 0, \end{aligned} \tag{7}$$

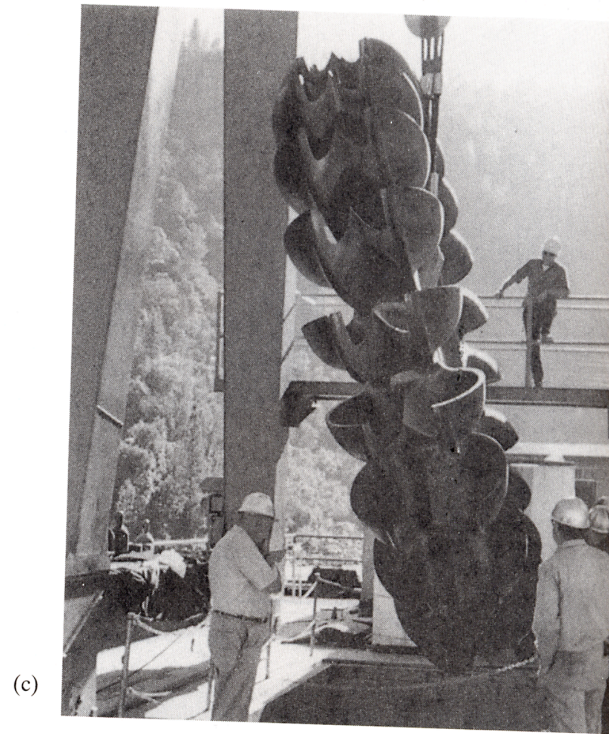
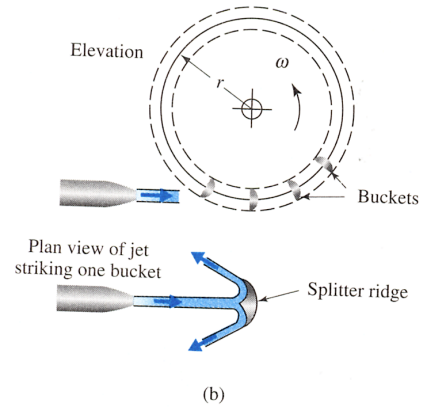
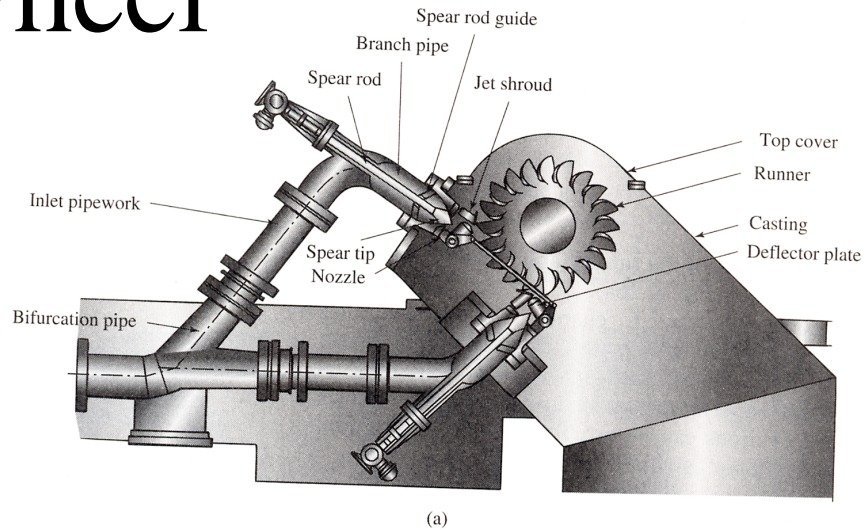
which *isn't* an ordinary differential equation in a single independent variable; in fact it's a lot more complicated, because we have a free y floating around. This means that the solution f to (7) will depend on y , which is inconsistent with (6), where we assumed that f was only a function of η . Thus (6) is *not* a valid solution of the differential equation (1). So you see that what we did in class to solve the problem wasn't quite so arbitrary after all, because if you guess a solution and it isn't valid, then the equations will tell you.

¹Things aren't quite this simple really – there are some complications with higher-order differential equations, etc. – but for us this statement is valid.

Turbines - water - power generation

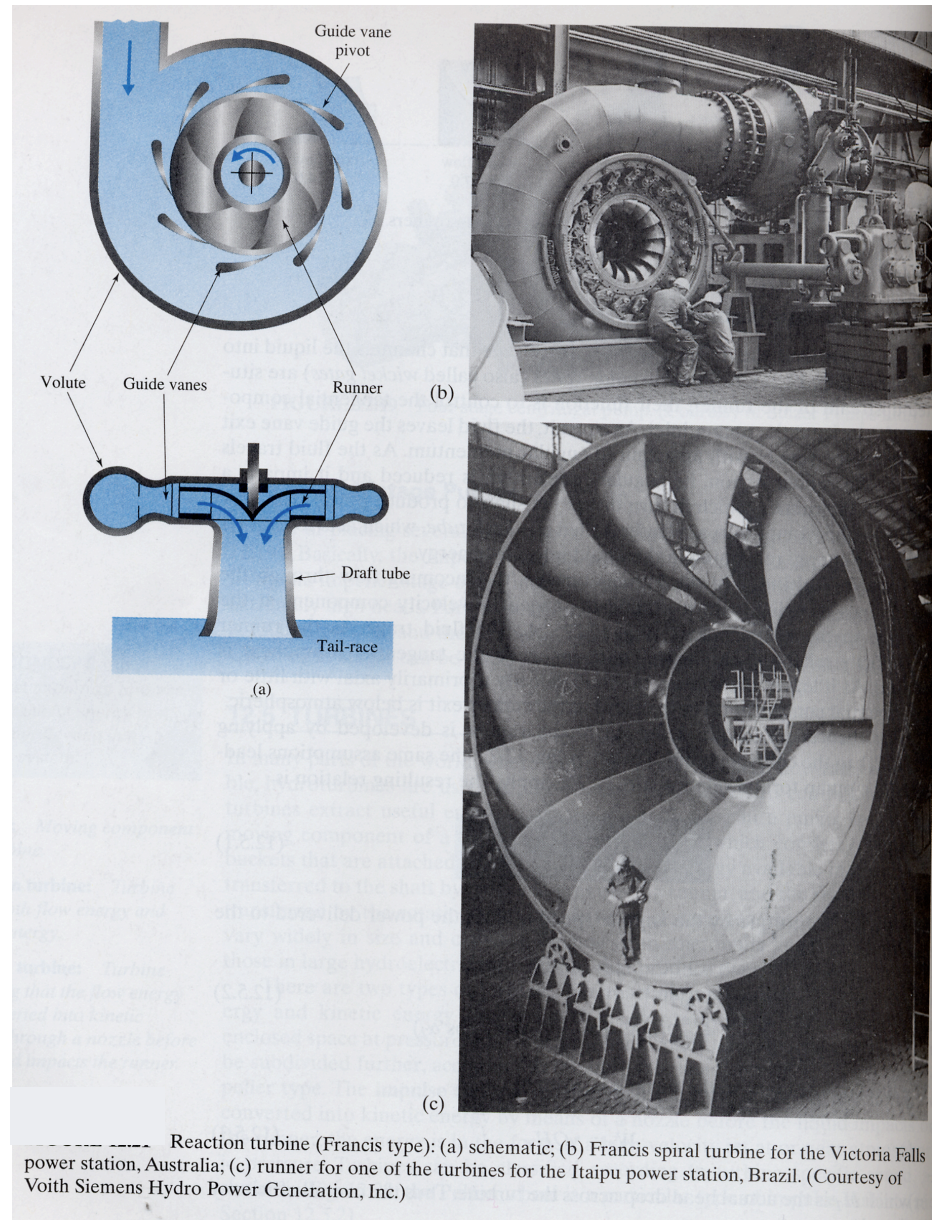
- Impulse Turbines - Pelton wheel
(kinetic energy only - nozzle - at p-atm)
- Reaction Turbines
(both kinetic energy and delta P - energy conversion takes place in enclosed space above p-atm)
 - Francis
 - Kaplan

Pelton wheel

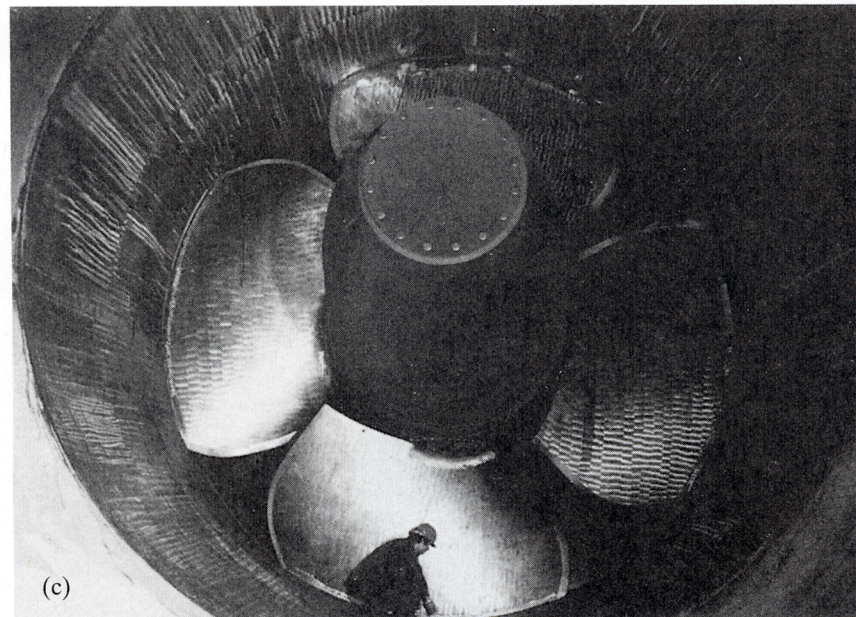
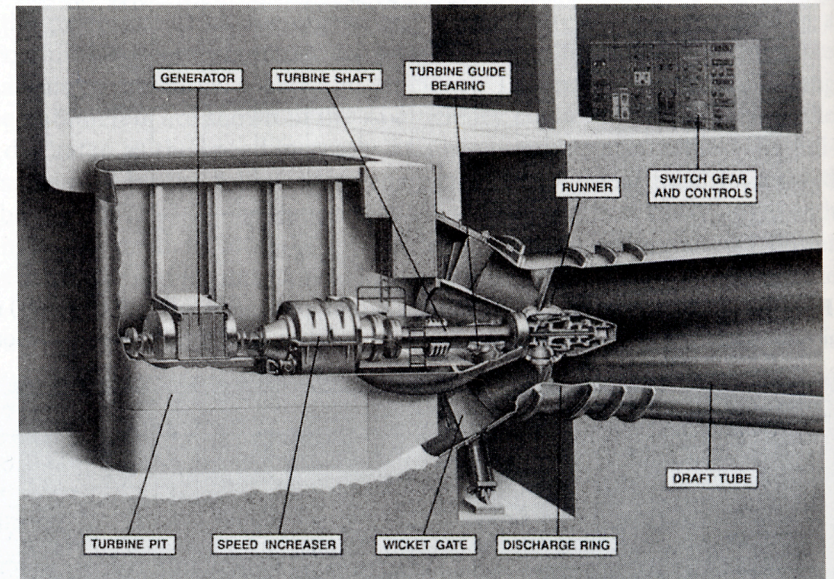
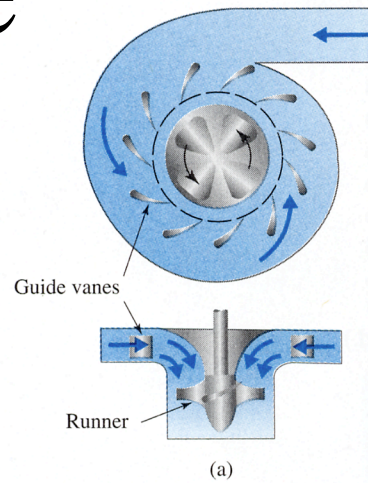


Impulse turbine (Pelton type): (a) typical arrangement of twin-jet machine. (Courtesy of Gilbert Gilkes and Gordon, Ltd.); (b) jet striking a bucket; (c) Pelton runner in the New Colgate power station, USA. (Courtesy Voith Siemens Hydro Power Generation, Inc.)

Francis-type reaction turbine

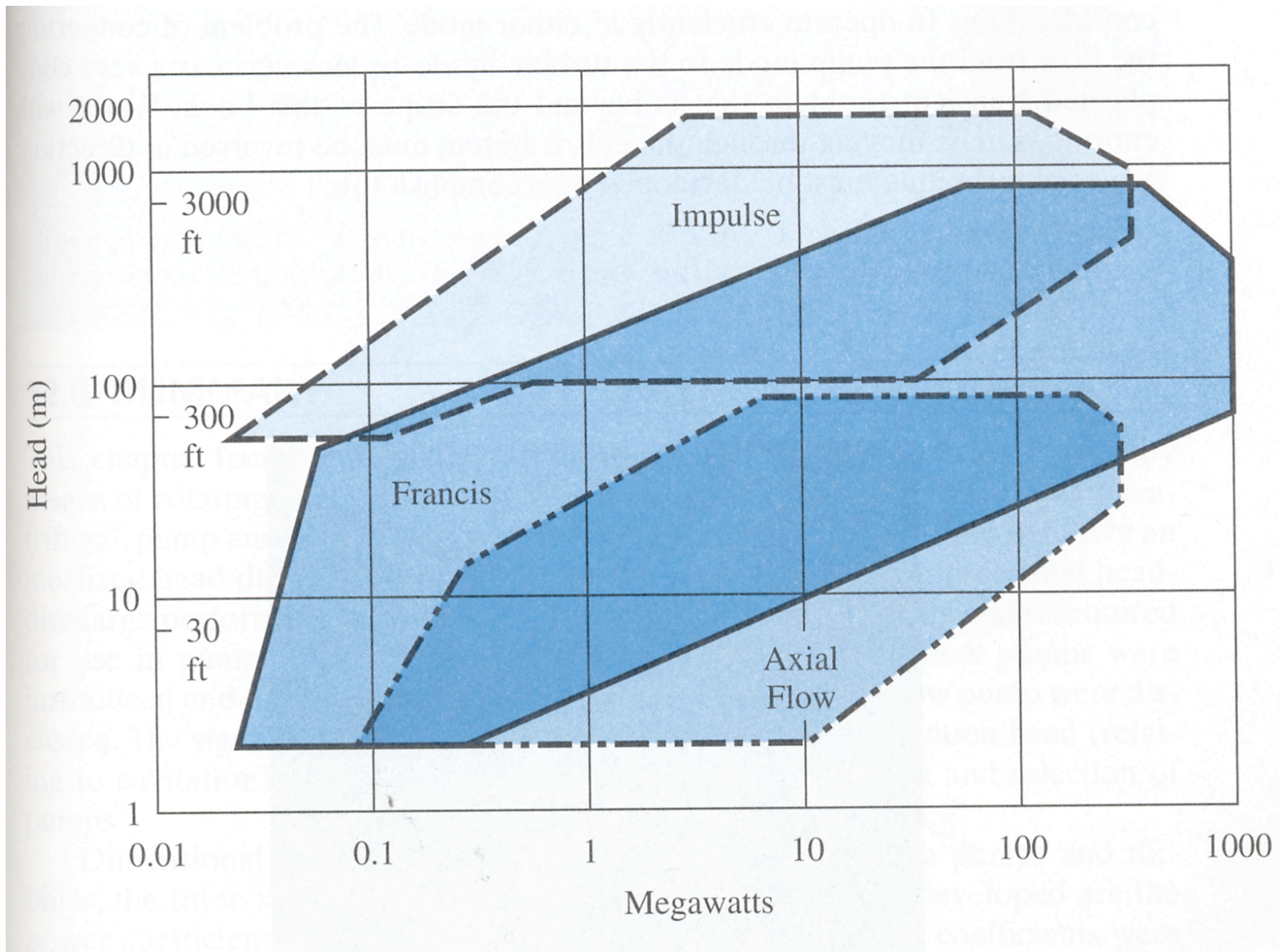


Kaplan-type reaction turbine



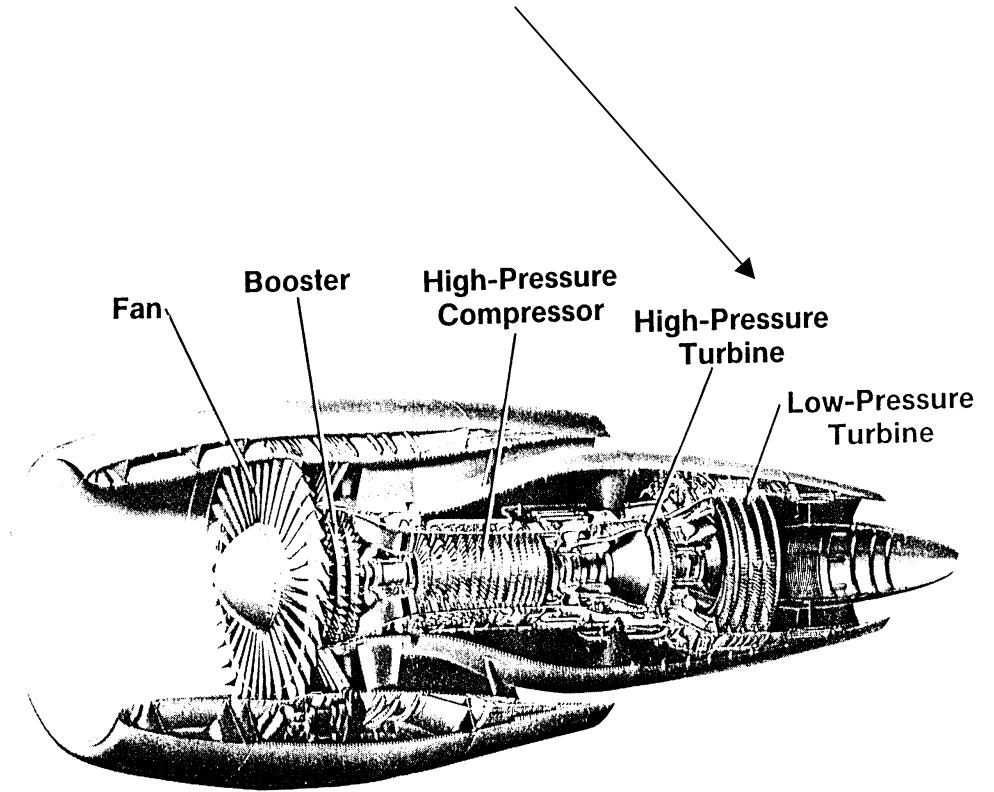
(b)

Axial-flow turbine
(Kaplan type): (a) schematic;
(b) side view; (c) turbine blades in
the Altenwörth power station,
Austria. (Courtesy of Voith Siemens
Hydro Power Generation, Inc.)



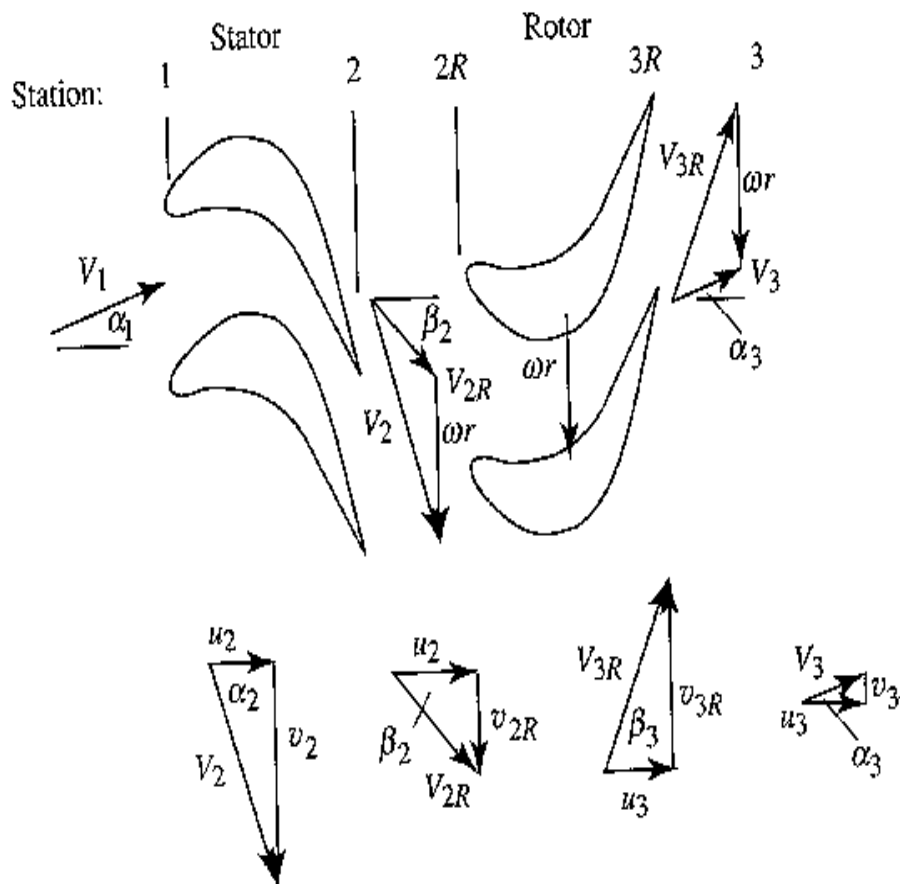
Jet engine: - gas turbine

General Electric CF6-50

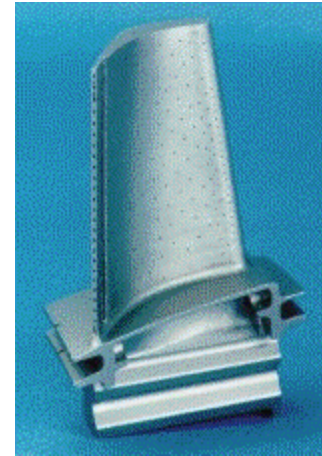


Gas turbine

ANALYSIS: Velocity Triangles

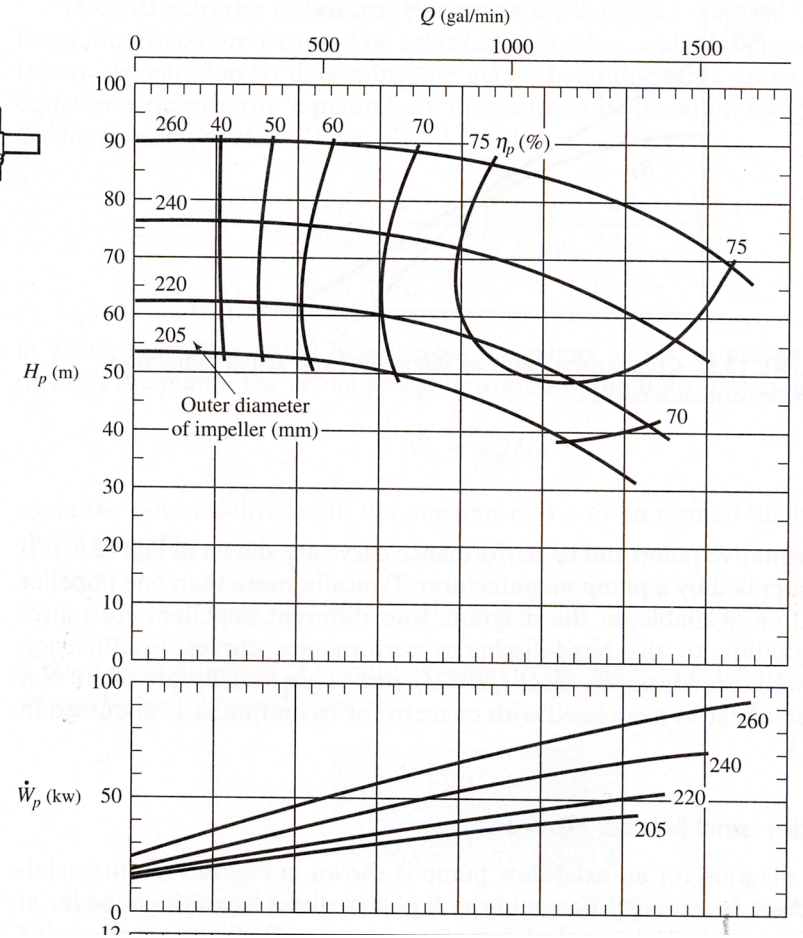
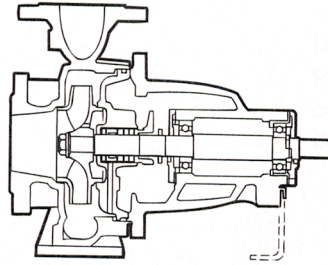


Thermal aspects: blade cooling

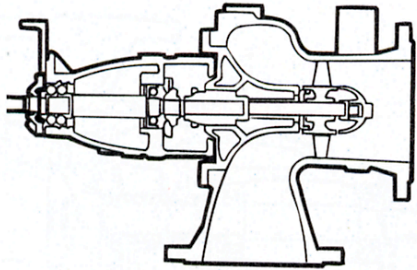


Pumps:

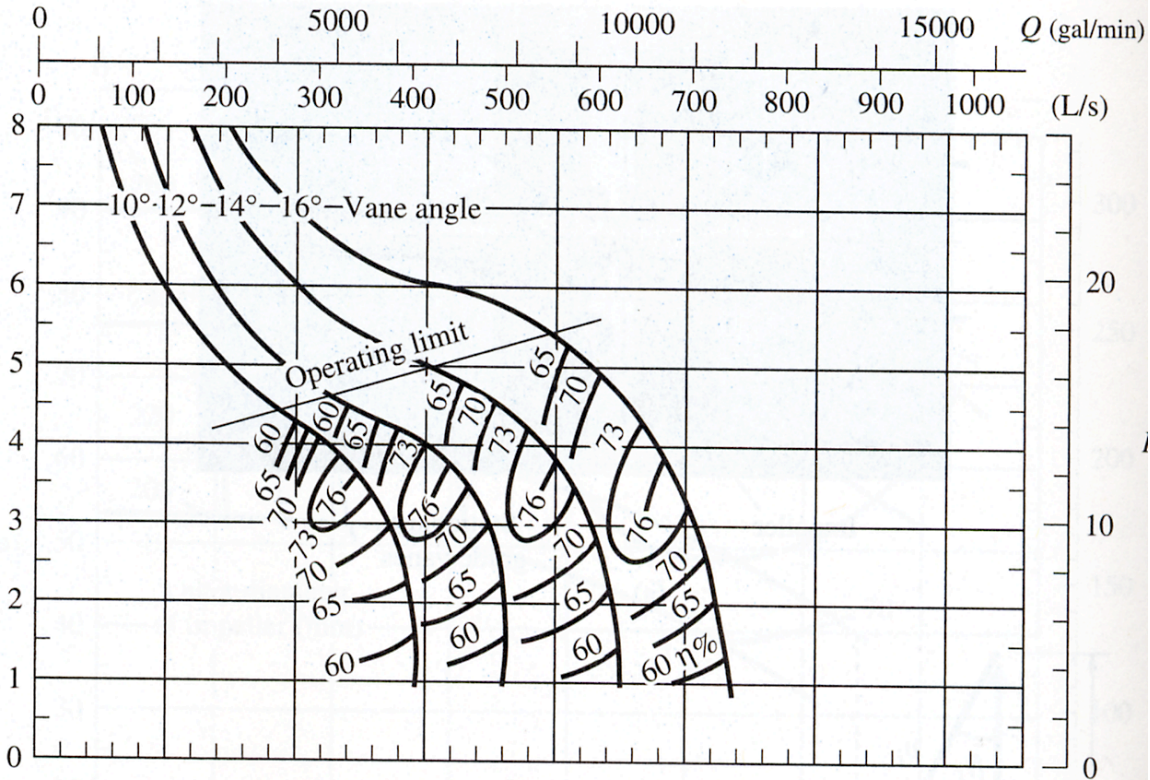
Centrifugal



Axial Pump

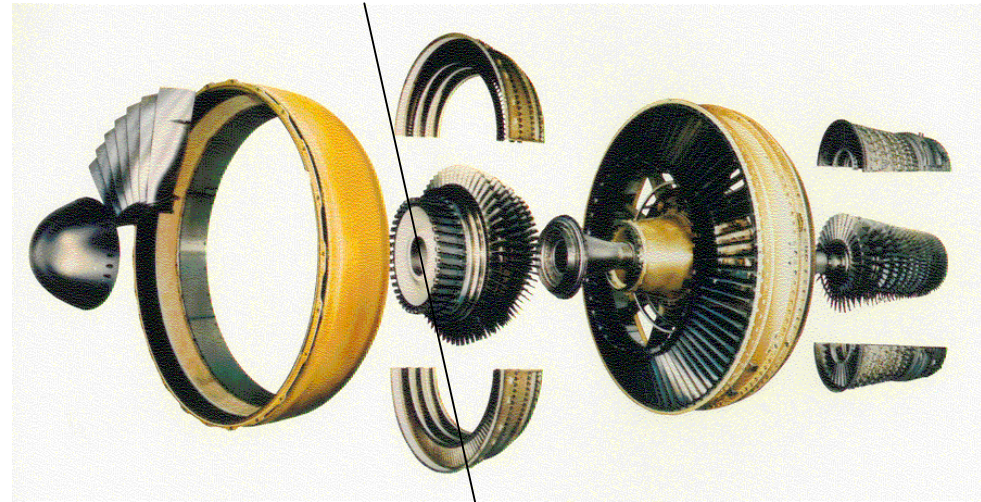


H_p (m)

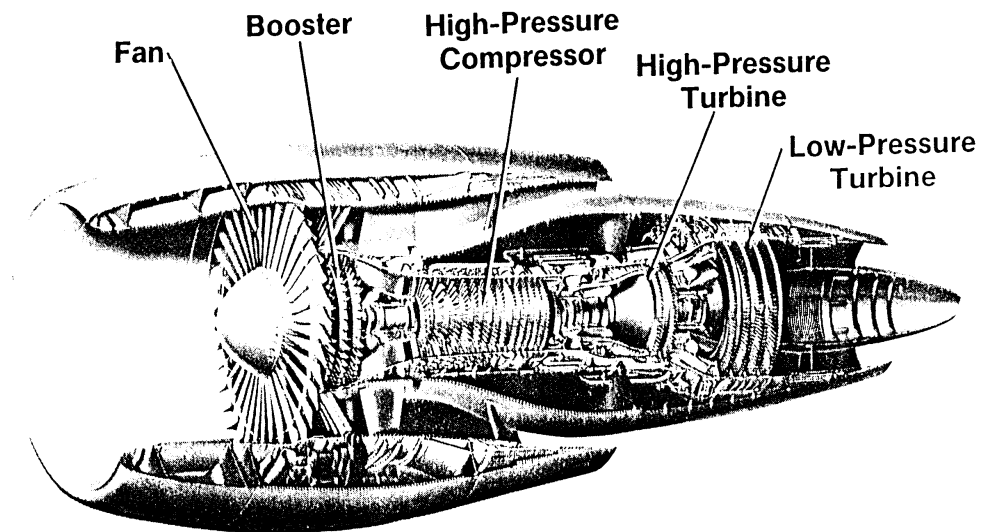


Compressors

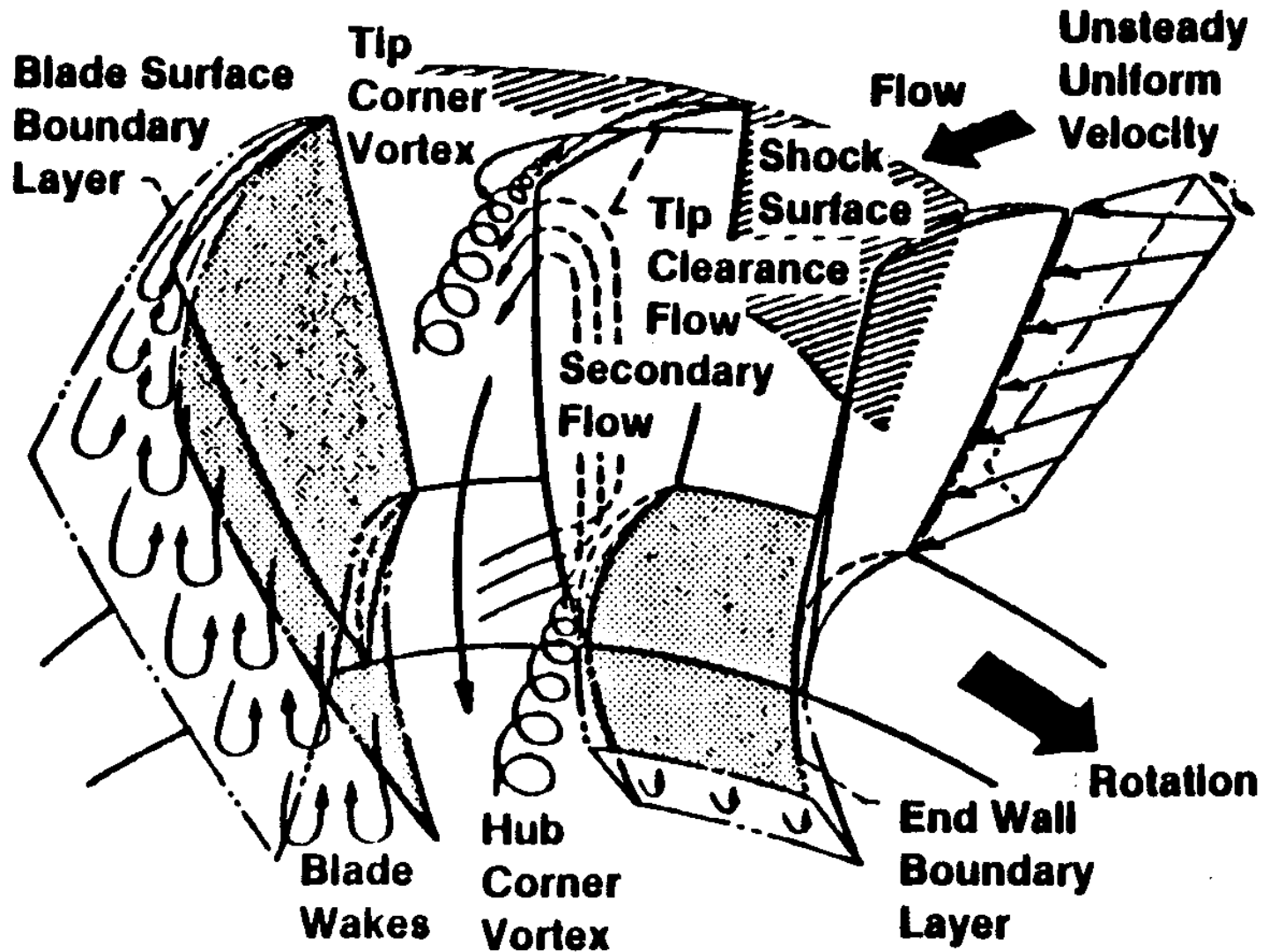
- AXIAL



General Electric CF6-50



Complex Flow in Compressors



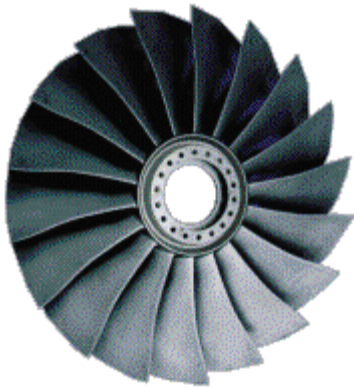
Fans



This 4-stage ATEGG Compressor Rig, the highest loaded compressor ever built with acceptable stall margin and efficiency characteristics, meets the Phase III stage loading goals.



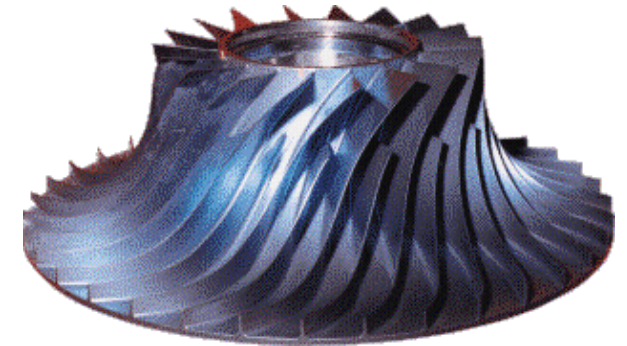
JTAGG III Advanced Concept Centrifugal Impeller with independent inducer and exducer will provide higher pressure ratio and efficiency.



JTDE Forward Swept Fan Blisk will be the first large forward swept rotor to be tested in a demonstrator engine.

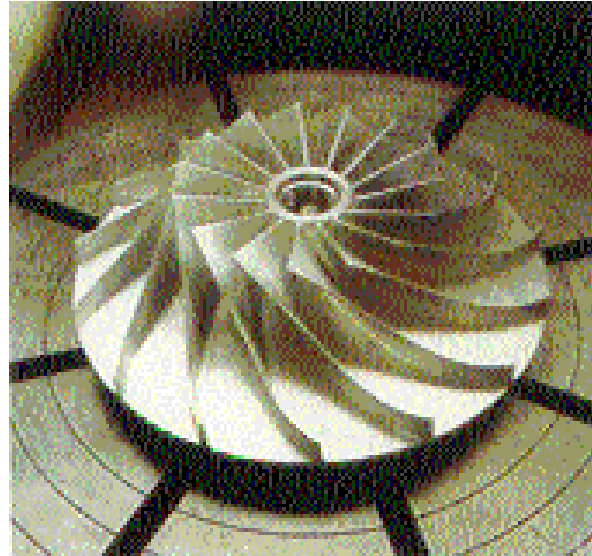


Integrating Forward Sweep and Splitter Technology are key features to achieving high efficiency and high pressure ratio. This stage will be utilized as the JTAGG III low pressure compressor.



A Forged Orthorhombic Transformed Super Alpha-2 Billet will be bonded to Gamma TiAl to form a novel, dual alloy impeller.

- Centrifugal compressors



Specific speed:

