Fluid Mechanics
EML 5709
Homework

Dr. Leon van Dommelen

Fall 2018

Do not print out this file. Keep checking for changes. Homeworks should normally be posted two lecture days before it is due. Deviations may occur due to holidays or the end of the semester.

Explain all reasoning.

Contents

1 9/05 W

1. If the density of air at sea level is 1.225 kg/m^3, and the molecular mass 28 g/mol, then what is the number of molecules per unit volume? What is the average spacing ℓ of the molecules?

   Consider a molecule of diameter d that moves over one free path length λ. During that motion it will hit another molecule if the center of the other molecule is within a radius d from the path of the molecule. In other words, the center of the other molecule must be inside a cylinder of radius d around the path λ of the first molecule. There should be about one collision in a free path, so there should be about one other molecule within the cylinder. So the free path can be ballparked from setting the volume of the cylinder equal to the average volume per particle:

   $$\pi d^2 \lambda = \text{average volume occupied per particle}$$

   Take the average diameter of the molecules to be 0.3 nm and compute λ. A more careful analysis says you still need to divide this by $\sqrt{2}$, so do so.

2. Suppose you have a body of typical size L. Which length, ℓ or λ, relative to L, determines whether you can define a meaningful (i.e. not nonsensical) "pointwise" density and velocity at a given time? (Here "pointwise" means using small volumes much smaller than L.) Which length determines whether you can define a meaningful "pointwise" density and velocity that would be enough info, say, to compute the further flow development? In particular, take sea-level air. Then for what body size L can you no longer use the normal (Navier-Stokes) equations to compute the flow at the "points" around the body? For what size body can you no longer find a meaningful continuum velocity for the "points" around the body even if you used molecular dynamics?

3. For ideal stagnation point as discussed in class, compute the pressure field (Eulerian) from the Bernoulli law. Then verify Newton's second law $\rho\vec{a} = -\nabla p$ (mass per unit volume times acceleration equals force per unit volume) using the Lagrangian expressions for the particle paths to get $\vec{a}$. Hints: Note that the pressure is a scalar, not a vector. And that you need to write it in terms of x and y, to take the gradient. The gradient of the pressure is a vector. The acceleration is the second derivative of the, Lagrangian, particle positions; read your notes.

4. If the surface temperature of a river is given by $T = 2x + 3y + ct$ and the surface water flows with a speed $\vec{v} = \hat{\imath} - \hat{\jmath}$, then what is c assuming that the water particles stay at the same temperature? (Hint: $\mathrm{D}T/\mathrm{D}t = 0$ if the water particles stay at the same temperature. Write this out mathematically.)

5. A boat is cornering through this river such that its position is given by $x_b = f_1(t)$, $y_b = f_2(t)$. What is the rate of change $\mathrm{d}T/\mathrm{d}t$ of the water temperature experienced by the boat in terms of the functions f_1 and f_2?

6. The velocity field of shallow water waves is near the surface given by

$$u = \epsilon \sin(kx + \omega t) \qquad v = -\epsilon \cos(kx + \omega t)$$

Find the pathlines for these water waves. Since this is a messy process, simplify it by assuming that ϵ is small. In that case the particle displacements are small, and that allows you to approximate x in the sine and cosine by the x-value ξ of the *initial* particle position, which is constant for a given particle:

$$u = \epsilon \sin(k\xi + \omega t) \qquad v = -\epsilon \cos(k\xi + \omega t)$$

Find and draw a representative collection of particle paths under that assumption.

2 9/12 W

1. As noted in the previous homework, the velocity field of shallow water waves is near the surface given by

$$u = \epsilon \sin(kx + \omega t) \qquad v = -\epsilon \cos(kx + \omega t)$$

where amplitude ϵ, wave number k, and frequency ω are all positive constants. Find and draw the streamlines of the flow. Do not assume epsilon is small in this case. Compare with the pathlines. Why are they not the same?

2. Draw the streakline coming from a generator at the origin, which is turned on at time $t = 0$. Draw the streakline for times $\omega t = 0,\ \frac{1}{4}\pi,\ \frac{1}{2}\pi,\ \pi,\ 2\pi,\ 4\pi,\ 6\pi,\ \ldots$. In a separate graph, draw the particle path of the particle that was released at time zero. Compare its position at the given times with that in the streaklines. Hint: Read up on how to get streaklines in your notes.

3. Substitute the Eulerian velocity field of stagnation point flow into the (Eulerian) Euler equations. In the force per unit volume, include the gravity force per unit volume. Assume that gravity is in the minus y-direction. You get three equations for the pressure, one giving its x-derivative, one its y derivative, and the third its z-derivative. More than one equation for a single scalar unknown p is usually too much, but show that in this case, there is indeed a solution p that satisfies all three equations. Find out what it is. Does it satisfy the Bernoulli law?

   Note: to find the pressure correctly, solve the Euler equation in the x-direction for the pressure. The integration constant will depend on y and z. Substitute this result into the Euler equation in the y-direction to narrow down the integration constant. Then substitute this result into the Euler equation in the z-direction to narrow down the constant even more.

3 9/19 W

1. In Poiseuille flow (laminar flow through a pipe), the velocity field is in cylindrical coordinates given by

$$\vec{v} = \widehat{\imath}_z v_{\max}\left(1 - \frac{r^2}{R^2}\right)$$

where $v_{\max}$ is the velocity on the centerline of the pipe and R the pipe radius. (a) Use Appendices B and C to find the velocity gradient and strain rate tensors of this flow. *Do not guess.* (b) Evaluate the strain rate tensor at $r = 0$, $\frac{1}{2}R$ and R. What can you say about the straining of small fluid particles on the axis? (c) Is Poisseuille flow an incompressible flow? (d) Also find the vorticity. Do particles on the axis rotate? If not, what *do* they do?

2. (20pt) For the Poisseuille flow of the previous question, derive the principal strain rates and the principal strain directions for an arbitrary radial position r using class procedure.

3. Make a neat picture of a vertical r, z plane through the axis showing, for a point at an arbitrary r, z, a small cubic fluid particle, in cross section, that is aligned with the principal strains at that point. Using arrows as appropriate, show the translational, straining, and average rotational motions that this particle is performing. If the particle was a small solid sphere, instead, would you expect it to rotate, and if so, in which direction?

4. Write out the continuity equation

$$\frac{1}{\rho}\frac{\mathrm{D}\rho}{\mathrm{D}t} + \mathrm{div}\vec{v} = 0$$

in cylindrical and spherical coordinates, for both a compressible and an incompressible fluid. (Here incompressible means that the density of individual fluid particles is constant, not that all fluid particles must have the same density. Usually, when people say incompressible they mean that the density is the same everywhere. But looking in the sea, different regions have different density, because of different salt, but the individual particles are still pretty much incompressible.) Note that

$$\frac{\mathrm{D}}{\mathrm{D}t} = \frac{\partial}{\partial t} + \vec{v}\cdot\nabla$$

and use the appendices.

Next assume that $\vec{v}$ and ρ only depend on r and t (so the flow is cylindrically or spherically symmetric). How do the equations simplify? In the incompressible case, you should see that there is a quantity that must be a constant (at least for any given time) in each flow. What is it? In the compressible *steady* flows, there is also a quantity that must be constant. What is that? What happens to the radial velocity when going to large r?

5. Use the expression derived in class to ballpark the kinematic viscosity ν of standard air. From that ballpark the kinematic viscosity μ. Use the data from Appendix A in the book and the posted solutions of homework 1 for the free path length. Compare with the exact values.

6. Two-dimensional Poiseuille flow (in a duct instead of a pipe) has the velocity field

$$\vec{v} = \hat{\imath} v_{\max} \left(1 - \frac{y^2}{h^2}\right)$$

Here x is along the centerline of the duct, y across the gap measured from the middle, and h is half the duct height. Neatly sketch the duct and its velocity profile. Find the viscous stress tensor for this flow, assuming a Newtonian fluid, using Table C3 in the book. Evaluate the viscous stress tensor at $y/h = \frac{1}{2}$. Draw a little cube of fluid at that position in the duct (in cross-section), and sketch all viscous stresses acting on that cube. In a different color, also sketch the inviscid pressure forces acting on it. (assume the pressure has some value p.)

Next assume that the little cube is rotated counter-clockwise over a 30 degree angle (around the z-axis). Find the total stresses σ (including pressure) normal and τ tangential on the now oblique front surface of the little cube. To do so, first find a unit vector $\vec{n}$ normal to the surface. Then find the vector stress on the surface using $\vec{R} = \bar{\bar{\tau}}\vec{n}$. Then find the components of $\vec{R}$ in the direction of $\vec{n}$ (so normal to the surface), and normal to $\vec{n}$ (so tangential to the surface).

Note: This is essentially question 5.3 from the book, but do not assume that the pressure is 5; just leave it as p.

4 9/26 W

1. The two-dimensional Poiseuille flow of the last homework had the velocity field

$$\vec{v} = \hat{\imath} v_{\max} \left(1 - \frac{y^2}{h^2}\right)$$

Here x is along the centerline of the duct, y across the gap measured from the middle, and h is half the duct height. Find the strain rate tensor of this flow, and from that the viscous stress tensor, assuming a Newtonian fluid. Compare with the direct expression for the stress tensor found in the last homework. Also write out the total stress tensor, (including pressure), as given by

$$T_{ij} = -p\delta_{ij} + \tau_{ij}$$

Here δ_{ij} is called the Kronecker delta or unit matrix, it is 1 if $i = j$ and zero otherwise.

2. 6.1. Use the appendices. Based on the results, discuss whether this is incompressible flow, and in what direction the viscous stresses on the surface of the sphere are. Also state in which direction the inviscid stress on the surface is.

3. Noting that in the above flow, the pressure is given by $p - p_\infty = -3\mu U\cos(\theta)r_0/2r^2$, evaluate the pressure and shear stresses on the surface. Then find and integrate the components of the pressure and viscous stresses in the axial direction (the line $\theta = 0$) on the surface of the sphere to find the viscous drag force on the sphere. Thus recover the Stokes formula for the drag of a sphere at low Reynold number, $F_\mathrm{D} = 6\pi\mu r_0 U$. Note that the surface element on a spherical surface of radius r_0 is given by $r_0^2 \sin\theta \mathrm{d}\theta\mathrm{d}\phi$.

4. 6.2 Discuss your result in view of the fact, as stated in (6.1), that the Reynolds number must be small for Stokes flow to be valid. So what about the dynamic pressure, (as produced by the kinetic energy of the fluid particles), in Stokes flow?

5. As seen in class, the second law requires that the dissipation for a Newtonian fluid may not be negative. Examine what constraints this puts on the values of μ and λ. To do so, first write out the strain rate tensor and then the compressible Newtonian stress tensor in terms of the strain rates *only*. (So write $\operatorname{div}\vec{v}$ in terms of the strain rates.) Then note that $\tau_{ij}s_{ij}$ simply means multiplying all corresponding components of the two tensors together and then adding all 9 terms together (much like taking a dot product between vectors). Then explain why μ must be positive (or at least not negative) because otherwise, say, a Couette flow field in which only $s_{12} = s_{21}$ is nonzero would violate the second law. Then argue that with μ positive, the worst-case scenario for negative entropy generation occurs when all off-diagonal ($i \neq j$) strain rates are zero. So you can from now on limit your considerations to only the terms involving diagonal ($i = j$) strain rates. (But that is expected, since you can always switch to principal axes where there are no off-diagonal terms.) For the diagonal terms the following trick works: your terms should include what can be considered the dot product between the vectors $\vec{v}_1 = (s_{11}, s_{22}, s_{33})$ and $\vec{v}_2 = (1, 1, 1)$. You should know that $\vec{v}_1 \cdot \vec{v}_2 = |\vec{v}_1||\vec{v}_2|\cos(\theta)$. Here $\cos^2(\theta)$ is no bigger than one, and it is one only if the two vectors are parallel. (In general this is known as the "Cauchy-Schwartz inequality.") From that argue that λ may not be more negative than $-\frac{2}{3}\mu$. In the marginal case of Stokes' hypothesis that λ *is* $-\frac{2}{3}\mu$, there is one particular straining in which the dissipation, though not negative, is zero. Show that that corresponds to a uniform expansion or compression in all directions. Apparently, such an expansion is perfectly reversible according to Stokes, unlike, say, a unidirectional expansion in the x-direction only. What do you think of that?

5 10/03 W

1. Write down the worked-out mathematical expressions for the integrals requested in question 5.1. This is a good exercise in identifying various surface and volume integrals in integral conservation laws. Explain their physical meaning, if any. Don't worry about actually doing the integrations. However, show integrands and limits completely worked out.

   Take the surfaces S_I, S_{II}, S_{III}, and S_{IV} to be one unit length in the z-direction. (To figure out the correct direction of the normal vector $\vec{n}$ at a given surface point, note that the control volume in this case is the right half of the region in between two cylinders of radii r_0 and R_0 and of unit length in the z-direction. The vector $\vec{n}$ is a unit normal vector sticking *out* of this control volume.)

2. 5.14. Find both the horizontal and vertical components of the force. Make sure that you clearly define what control volume you are using, as there is no unique choice.

3. 5.11. Clearly define what control volume you are using.

6 10/10 W

Hurricane Michael.

7 10/17 W

1. 5.12. This question explains why the water stream coming out of a faucet contracts in area immediately below the faucet exit. As always, both mass and momentum conservation are needed.

   The faucet exit velocity may be assumed to be of the form of Poisseuille flow:

   $$v_z = v_{\max}\left(1 - \frac{r^2}{R^2}\right)$$

   You can assume that the stress tensor at the faucet exit is of the form (in cylindrical coordinates)

   $$\bar{\bar{\tau}} = \begin{pmatrix} 0 & 0 & \tau_0 r/R \\ 0 & 0 & 0 \\ \tau_0 r/R & 0 & 0 \end{pmatrix}$$

   in other words, much like the strain rate tensor that you derived earlier for Poisseuille flow.

   Take the faucet exit as the entrance of your control volume. Take as exit to your control volume a slighly lower plane at which the radius of the jet has stabilized to R_2 and the flow velocity has become uniform (independent of r). For a uniform flow velocity there are no viscous stresses. Gravity can be ignored compared to the high viscous forces in this very viscous fluid. (However, over a longer distance gravity will lead to a further thinning of the jet.) And you can assume that the pressure at the exit is already atmospheric, as it definitely is in the lower plane below.

2. Write a finite volume discretization for the x-momentum equation for the little finite volume in polar coordinates. Just like the continuity equation done in class, your final equation should *only* involve pressures, densities, and velocities at the center points of the finite volumes. Ignore the viscous stresses for now.

The unknown velocities used in the computation should be taken to be the polar components v_r and v_θ. But momentum conservation for x-momentum is asked. (Conservation of r-momentum or θ-momentum would be complete nonsense.) So you will need to write the x-component of velocity in terms of the polar unknowns. Note that in Cartesian coordinates, the polar unit vectors are given by

$$\widehat{\imath}_r = \cos(\theta)\widehat{\imath} + \sin(\theta)\widehat{\jmath} \qquad \widehat{\imath}_\theta = -\sin(\theta)\widehat{\imath} + \cos(\theta)\widehat{\jmath}$$

These should be able to allow you to evaluate the x-components of velocity and pressure forces that you need.

3. Assuming that there are known viscous stresses at the centers of the sides of the finite element, what additional terms do you get in the obtained equation due to viscous forces? Assume the stress tensor is given in polar form. (So τ_{rr}, $\tau_{r\theta}$, etcetera.) Once again you will need the polar unit vectors to get x-components of the forces.

8 10/24 W

1. 7.5. Use the appendices. You may only assume that $v_r = v_r(r)$, $v_\theta = v_\theta(r)$, $v_z = 0$, and $p = p(r, \theta)$ in cylindrical coordinates. (And that the fluid is Newtonian with constant density and viscosity, of course.) Do not assume that the radial velocity is zero, derive it. Do not assume that the pressure is independent of θ, derive it. Ignore gravity as the question says. Note that p must have the same value at $\theta = 0$ and 2π because physically it is the same point. Answer for v_θ:

$$\frac{\Omega r_0^2 r_1}{r_1^2 - r_0^2}\left(\frac{r_1}{r} - \frac{r}{r_1}\right)$$

2. In 7.5, what is the moment needed to keep the rod rotating, per unit axial length? What is the power needed? What is the pressure difference between the surfaces of the pipe and the rod?

3. 7.6. Do not ignore gravity, but assume the pipe is horizontal. And that the y-axis of the Cartesian coordinate system with z along the axis of the pipe is pointing upwards. Careful, the gravity vector is *not* constant in polar coordinates. Find the components using geometry or from $\vec{g} = -g\nabla h$. Assume x is horizontal like z and y vertically upwards. Do not ignore the pressure gradients: assume the pressure can be any function $p = p(r, \theta, z, t)$ and derive anything else. Merely assume that the pressure distribution at the end of the pipe and rod combination is the same as the one at the start. For the velocity assume $v_r = v_\theta = 0$ and $v_z = v_z(r, z)$. Anything else must be derived. Give both velocity and pressure field. Check that your answer is the same as you would get from using a kinetic pressure. What is the force required to pull the rod through the axis, per unit length?

4. Consider the below graph for the minor head losses due to sudden changes in pipe diameter:

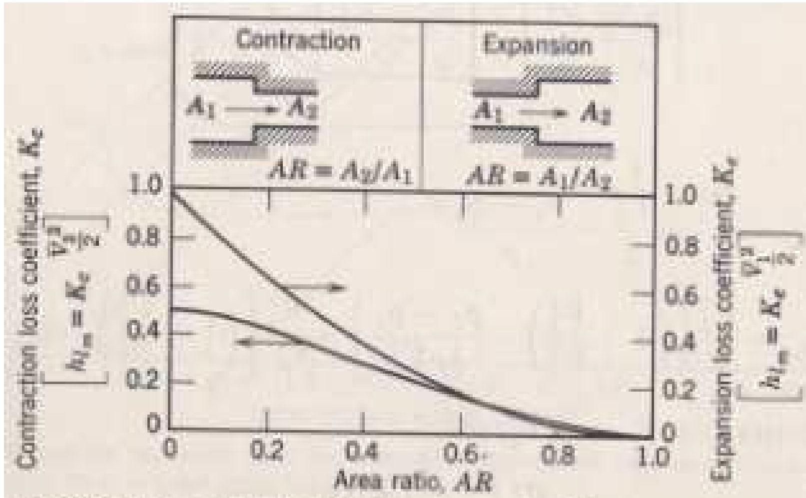

Fig. 8.16 Loss coefficients for flow through sudden area changes (Ref. 5).

Discuss the following issues as well as possible from the sort of flow you would expect.

(a) How come this minor head loss coefficient becomes zero for an area ratio equal to 1?

(b) Why do they use different scales and reference velocities for a sudden contraction than for a sudden expansion?

(c) Why would the head loss coefficient be exactly one for a large expansion? Coincidence?

(d) Why would the head loss coefficient be less than one if the expansion is less? If the expansion is less, is not the pipe wall in the expanded pipe closer to the flow, so should the friction with the wall not be more??

(e) Why is there a head loss coefficient for a sudden contraction? The mechanism cannot be the same as for the sudden expansion, surely? Or can it?

(f) Any other observations you can offer?

In answering this, think of where the head loss comes from, what its source is. What is lost?

9 10/31 W

1. (Part of 7.17 with $n = 1$ only.) Assume that an infinite flat plate normal to $\widehat{\jmath}$ accelerates from rest, so that its velocity is given by $u_{\text{plate}}\,\widehat{\imath} = \dot{U}t\,\widehat{\imath}$ where $\dot{U}$ is a constant. There is a viscous Newtonian fluid above the plate. Assuming only that $\vec{v} = \vec{v}(y,t)$, $w = 0$, and that the effective pressure far above the plate is constant, derive a partial differential equation and boundary conditions for the flow velocity of the viscous fluid. List them in the plane of the independent variables.

2. (Part of 7.17 with $n = 1$ only.) Use dimensional analysis to show that the fluid velocity profile is similar,

$$\frac{u}{\dot{U}t} = f(\eta) \qquad \eta = \frac{y}{\sqrt{4\nu t}}$$

(The units of the constant $\dot{U}$ should be obvious.) Hint: use $\dot{U}$ and t as repeating parameters. Then, based on the above expression for u, and the appropriate equation of the previous question, work out the equation that the scaled velocity profile f has to satisfy.

3. (Part of 7.17 with $n = 1$ only.) Find the solution for the velocity profile from the equation found in the previous question. One way to do so is differentiate the equation for f twice with respect to η, and so show that $g = f''$ satisfies the equation

$$g'' + 2\eta g' = 0$$

This equation is the same as the one for f in Stokes' second problem, and was solved in class. The general solution was

$$g(\eta) = C_1 \int_{\bar{\eta}=\eta}^{\infty} e^{-\bar{\eta}^2}\,\mathrm{d}\bar{\eta} + C_2$$

Explain why C_2 must be zero. Explain why then f' can be found as

$$f'(\eta) = -\int_{\bar{\eta}=\eta}^{\infty} g(\bar{\eta})\,\mathrm{d}\bar{\eta} = -C_1 \int_{\bar{\eta}=\eta}^{\infty} \int_{\bar{\bar{\eta}}=\bar{\eta}}^{\infty} e^{-\bar{\bar{\eta}}^2}\,\mathrm{d}\bar{\bar{\eta}}\,\mathrm{d}\bar{\eta}$$

Draw the region of integration in the $\bar{\eta}, \bar{\bar{\eta}}$-plane. Use the picture to change the order of integration in the multiple integral and integrate $\bar{\eta}$ out. Show that

$$f'(\eta) = C_1 \left[\eta \int_{\bar{\bar{\eta}}=\eta}^{\infty} e^{-\bar{\bar{\eta}}^2}\,\mathrm{d}\bar{\bar{\eta}} - \frac{1}{2} e^{-\eta^2} \right]$$

Integrate once more to find $f(\eta)$. Apply the boundary condition to find C_1.

Another way to solve is find the solution in a suitable math handbook. Note from the above that the solution is related to the error function somehow. Unfortunately, basic handbooks may not have the solution. You may need a somewhat more advanced book, like Abramowitz and Stegun.

10 11/07 W

1. For an ideal point vortex at the origin, the velocity field is given in cylindrical coordinates r, θ, z by

$$\vec{v} = \frac{\Gamma}{2\pi r} \hat{\imath}_\theta$$

Show that the vorticity $\vec{\omega} = \nabla \times \vec{v}$ of this flow is everywhere zero. Now sketch a contour (closed curve) C that loops *once* around the vortex at the origin, in the counter-clockwise direction. In fluid mechanics, (for *any* flow, not just this one), the "circulation" $\bar{\Gamma}$ of a contour is *defined* as

$$\bar{\Gamma} = \oint_C \vec{v} \cdot \mathrm{d}\vec{r}$$

Here the integration starts from an arbitrary point on the contour and loops back to that point in the counter-clocwise direction. Evaluate the circulation of your contour around the vortex. *Do not take a circle as contour C; take a square or a triangle or an arbitrary curve.* Of course you know that in polar coordinates an infinitesimal change $\mathrm{d}\vec{r}$ in position is given by

$$\mathrm{d}\vec{r} = \widehat{\imath}_r \mathrm{d}r + \widehat{\imath}_\theta r \mathrm{d}\theta$$

(If not, you better also figure out what it is in spherical.) You should find that Γ has a nonzero value for your contour.

2. So far so good. But the Stokes theorem of Calculus III says

$$\oint \vec{v} \cdot \mathrm{d}\vec{r} = \int_A \nabla \times \vec{v} \cdot \vec{n}\, \mathrm{d}A$$

where A is an area bounded by contour C. You just showed that the left hand side in this equation is *not* zero, but that the right hand side is because $\nabla \times \vec{v}$ is. Something is horribly wrong???! To figure out what is going on, instead of using an ideal vortex, use the "Oseen vortex"

$$\vec{v} = \frac{\Gamma}{2\pi r}\left(1 - e^{-r^2/4\nu t}\right)\widehat{\imath}_\theta$$

To simplify the integrations, now take your contour C to be (the perimeter of) a circle around the origin in the x, y-plane, and take area A to be the inside of that circle in the x, y-plane. Do both the contour integral and the area integral. In this case, they should indeed be equal. Now in the limit $t \downarrow 0^+$, the Oseen vortex becomes an ideal vortex (the exponential becomes zero). (The Oseen vortex is an *initially* ideal vortex that diffuses out in time due to viscosity.) So if you look at a very small time, you should be able to figure out what goes wrong for the ideal vortex with the Stokes theorem. You might want to plot the vorticity versus r for a few times that become smaller and smaller. Based on that, explain what goes wrong for $t \downarrow 0^+$. Is the area integral of the vorticity of the ideal vortex really zero? Read up on delta functions.

3. Do bathtub vortices have opposite spin in the southern hemisphere as they have in the northern one? Derive some ballpark number for the exit speed and angular velocity of a bathtub vortex at the north pole and one at the south pole, assuming that the bath water is initially at rest compared to the rotating earth. Use Kelvin's theorem. Note that the theorem applies to an inertial frame, not that of the rotating earth. So assume you look at the entire thing from a passing star ship. (Since you cannot see through the earth, you will either need to fly above the north pole or above the south pole, seeing different directions of rotation of the earth, counter-clockwise respectively clockwise.) What do you conclude about the starting question? In particular, how do you explain the bathtub vortices that we observe?

4. A Boeng 747 has a maximum take-off weight of about 400,000 kg and take-off speed of about 75 m/s. The wing span is 65 m. Estimate the circulation around the wing from the Kutta-Joukowski relation. This same circulation is around the trailing wingtip vortices. From that, ballpark the typical circulatory velocities around the trailing vortices, assuming that they have maybe a diameter of a quarter of the span. Compare to the typical take-off speed of a Cessna 52, 50 mph.

5. Model the two trailing vortices of a plane as two-dimensional point vortices (three-dimensional line vortices). Take them to be a distance 2ℓ apart, and to be a height h above the ground. Take the ground as the x-axis, and take the y-axis to be the symmetry axis midway between the vortices. Now:

   (a) Identify the mirror vortices that represent the effect of the ground on the flow field. Make a picture of the x, y-plane with all vortices and their directions of circulation.

   (b) Find the velocity at an arbitrary point x on the ground due to all the vortices.

   (c) Also find the velocity that the right-hand non-mirror vortex R experiences due to the other vortices. In particular find the Cartesian velocity components $u_{\rm R}$ and $v_{\rm R}$ in terms of Γ, h and ℓ.

6. Continuing the previous question, the right non-mirror vortex R moves with the velocity that the other vortices induce:

$$\frac{{\rm d}\ell}{{\rm d}t} = u_{\rm R} \qquad \frac{{\rm d}h}{{\rm d}t} = v_{\rm R}$$

   If you substitute in the found velocities and take a ratio to get rid of time, you get an expression for ${\rm d}h/{\rm d}\ell$. Integrate that expression using separation of variables to find the trajectory of the vortices with time. Accurately draw these trajectories in the x, y-plane, indicating any asymptotes. Do the vortices end up at the ground for infinite time, or do they stay a finite distance above it?

11 11/14 W

1. Find the streamfunction for ideal flow around a circular cylinder where the incoming flow at large distances has velocity $\vec{v} = U\hat{\imath}$ with U a constant. To do so, first verify that $r^{\pm n}\cos(n\theta))$ and $r^{\pm n}\sin(n\theta))$ are solutions of the Laplace equation by plugging them into the Laplacian in polar coordinates. Next find the streamfunction that describes the $U\hat{\imath}$ flow at large distances in Cartesian coordinates, and convert it to polar coordinates. Next add a multiple of a $r^{-n}\cos(n\theta))$ or $r^{-n}\sin(n\theta))$ term (with $n > 0$ so that the flow at large r is not affected) to satisfy the appropriate boundary condition at the cylinder surface $r = a$.

2. You should have found the streamfunction to be

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

Find the polar velocity components on the surface of the cylinder from this streamfunction. (Note that appendix D.2 has an error; the correct equation is $v_\theta = -\partial\psi/\partial r$.) Is the velocity normal to the surface zero as it should be? Is the velocity tangential to the surface the same as we got from the velocity potential? Use the Bernoulli law to find the pressure on the surface in terms of the pressure p_∞ far upstream. Where is the pressure on the surface p_∞? Where do you have stagnation pressure on the surface?

3. The streamfunction of an ideal vortex at the origin equals $(\Gamma/2\pi)\ln r$. Show that this produces $v_r = 0$ and $v_\theta = -\Gamma/2\pi r$. Add this to the streamfunction of the cylinder, above. Show that the velocity component normal to the surface is still zero. So we now have a cylinder with circulation around it. Recompute the pressure on the surface. Then integrate the pressure forces on the surface to find the net horizontal and vertical forces on the cylinder. According to D'Alembert, you should find that the horizontal force (the drag) is zero in this ideal flow. Is it? According to Kutta-Joukowski, you should find that the vertical force (the lift) is $\rho U \Gamma$. Is it?

4. Videos Dynamics: Potential flows: 290-294 and 299. In 290-293, do not try to accurately reproduce the body sizes. Just take the absolute values of the singularity strengths 2. And in 292, space them 4 intervals apart. In 294, just show the streamlines of the target flow, and comment on why one singularity is weird. In 299, try to match the experimental flow reasonably well. Note in doing so that you can put new singularities on top of old ones to adjust their strength; you do not have to start again from scratch. Alt-PrintScreen should send the plots to the clipboard, so that you can paste it into a program like MS Paint, where you can save it and print it out. In all cases, shade or highlight the part of the flow field that you would want to "solidify" (replace by a solid body like a cylinder or whatever). To get the body contour accurately, starting streamlines from near the stagnation points can be effective.

5. Go to the class airfoil programs[1] page. Download Matlab program `cylinder.m`. Print it out and in the print-out mark where the complex potential for flow around a circular cylinder is being set. Also mark where the streamlines are being drawn, and how that works. Run the program in Matlab and print out the streamlines around a circular cylinder. Then set variable Γ (`Gamma`) to an interesting value and print out those streamlines too.

12 11/28 W

1. Go to the class airfoil programs[2] page. Download Matlab program `airfoil.m`. Mark where a cylinder potential is set in a complex ζ-plane. Also mark where this cylinder is mapped to a Joukowski airfoil in a complex z-plane, and list the formula used to get z from ζ that achieves this mapping. You may observe that program `airfoil.m` is astonishingly simple for the complexity of the flow and graphics that it produces.

[1] `https://www.eng.famu.fsu.edu/~dommelen/courses/flm/progs/jou_air/`

[2] `https://www.eng.famu.fsu.edu/~dommelen/courses/flm/progs/jou_air/`

Then in Matlab, select parameters that produce the flow around a slightly cambered Joukowski airfoil of roughly 10% thickness ratio at 15 degrees angle of attack. List your parameters. Plot out the picture of the streamlines and isobars. Next set the circulation to zero by setting variables `Gamma` and `auto` both to zero and replot to show the effect of not satisfying the Kutta condition. Do the streamlines still come off smoothly from the trailing edge? What happens to the pressure?

2. Streamlines around a cylinder or sphere for very low Reynolds number, very *viscous* Stokes flow look superficially the same as those for high Reynolds number ideal *in*viscid flows: both are symmetric front/rear. But do they really look the same? Find out. Unfortunately, Stokes flow around a cylinder is tricky. For truly low Reynolds numbers, the flow in the vicinity of the cylinder becomes negligible, so the streamlines become infinitely widely spaced. But Stokes flow around a sphere is more reasonable.

   So, use the Matlab program streamlines.m[3] to first plot the streamlines for potential flow around a cylinder. Then create similar programs to plot the streamlines for both potential and Stokes flow around a sphere. You will need to put in the appropriate streamfunctions at the mesh points; potential flow around a sphere is in section 19.8, and Stokes flow in 21.8. You also need to change the values of the streamfunction that are plotted as streamlines; your streamlines should be equally spaced far upstream. The simplest way to do so is write the axysymmetric streamfunction far upstream, and then figure out what streamfunction values over there correspond to y values equal to $\pm 0.5\Delta y, \pm 1.5\Delta y, \ldots$. Comment on the differences in streamline spacing between potential and Stokes flow near the surface. Also comment on the differences in streamline curvature one or two radii away from the sphere.

3. Compute approximate values of the Reynolds number of the following flows:

   (a) your car, assuming it drives;

   (b) a passenger plane flying somewhat below the speed of sound (assume an aerodynamic chord of 30 ft);

   (c) flow in a 1 cm water pipe if it comes out of the faucet at .5 m/s,

   In the last example, how fast would it come out if the Reynolds number is 1? How fast at the transition from laminar to turbulent flow?

4. Using suitable neat graphics, show that the boundary layer variables for the boundary layer around a circular cylinder of radius a in a cross flow with velocity at infinity equal to U and pressure at infinity p_∞ are given by:

$$x = a\theta \qquad y = r - a \qquad u = v_\theta \qquad v = v_r$$

   Write the three appropriate partial differential equations for the unsteady boundary layer flow around a circular cylinder in terms of the *boundary layer variables* above. Also write the boundary conditions at the wall and above the boundary layer, at

[3] `https://www.eng.famu.fsu.edu/~dommelen/courses/flm/18/hw/streamlines.m`

$y/\sqrt{\nu} \approx \infty$. (Remember that $y/\sqrt{\nu} \approx \infty$ in the boundary layer solution should be taken to be equivalent to $y \approx 0$ in the potential flow above it.) Assume an unsteady flow impulsively started from rest, where you can assume that outside the thin boundary layer, the flow is still given by the ideal flow solution. Solve the pressure field inside the boundary layer fully.

5. Rewrite the exact Navier-Stokes equations in polar coordinates, (the continuity equation and the r and θ momentum equations) in terms of the *boundary layer* variables x, y, u, and v and the radius of the cylinder a. (So r, θ, v_r and v_θ may no longer appear in the equations.) Carefully distinguish between r (which may not appear in the results) and a. Compare these exact equations with the boundary layer equations. Explain for each discrepancy why the difference is small at high Reynolds numbers, where the boundary layer is thin.

13 12/05 M

1. According to potential flow theory, what would be the lift per unit span of a flat-plate airfoil of chord 2 m moving at 30 m/s at sea level at an angle of attack of 10 degrees? What would be the drag?

   Next, what would be the viscous drag if you compute it as if the airfoil is a flat plate aligned with the flow with that chord and the flow is laminar? Only include the shear stress over the last 98% of the chord, since near the leading edge the shear stress will be much different from an aligned flat plate. What is the lift to drag ratio? Comment on the value. Use $\rho = 1.225$ kg/m^3 and $\nu = 14.5\ 10^{-6}$ m^2/s.

2. Assume that a flow enters a two dimensional duct of constant area. If no boundary layers developed along the wall, the centerline velocity of the flow would stay constant. Assuming that a Blasius boundary layer develops along each wall, what is the correct expression for the centerline velocity?

3. Continuing the previous question. Approximate the Blasius velocity profile to be parabolic up to $\eta = 3$, and constant from there on. At what point along the duct would you estimate that developed flow starts based on that approximation? Sketch the velocity profile at this point, as well as at the start of the duct and at the point of the duct where the range $0 \leq \eta \leq 3$ corresponds to $\frac{1}{8}$ of the duct height accurately in a single graph. Remember the previous question while doing this!

14 12/07 M

1. For the turbulent mixing layer where the fluid at one side is at rest, give and plot the theoretical half shear layer thickness versus x. Use data in the book for any constants. Do the flows shown in the pictures seem consistent with the obtained relationship?

2. For the plane and axisymmetric turbulent jets, give and plot the half jet thickness versus x. Get any constant from the book. Also plot the maximum jet velocity versus

x for the jets. The book says somewhere that the axisymmetric (round) jet has a faster centerline (maximum) velocity decay as the two-dimensional jet. How did they get that crazy idea?

3. For the plane and axisymmetric turbulent wakes, plot the wake thickness versus x. How do they compare? Also plot the maximum velocity defects versus x and compare those. What is the problem with the axisymmetric turbulent wake for large enough x?

4. Would the similarity arguments made in class for a turbulent jet hold for a jet in a coflowing stream (like a jet coming out of a jet engine of a plane in flight)? If not, why not?