SBS, Design of Organisms: Momentum, Mass and Information Transfer 06 November 2006

Comments on Midterm I

1. Most of you correctly identified the terms in the equation for ***Re*** and their dimensions. We discussed several specific types of ***Re:*** body, roughness, local, pipe, and channel ***Re.*** Nobody chose to be even more specific, but they could have been. For example, you might choose the body ***Re*** for either a sphere or for a cylinder in cross flow; they are not identical.

   For a body ***Re***, you are correct that the usual convention is to choose the longest body dimension parallel to the flow. This convention (or any choice) presents no problems as long as the shape and orientation of the body to the flow both remain the same, as they must to maintain the same ***Re.*** For reasonably simple shapes, however, I would argue that it makes more sense to focus on the *smaller* of the two axes *perpendicular* to the flow. As long as the cylinder is longer than its diameter, for example, the diameter and not the length dominates its effects on the pattern of flow. Flows react most to how hard it is to get around an object in the flow, and they take the easiest route. The velocity to measure would be the free-stream velocity *upstream* of the object or more generally the difference in velocity between the object and the fluid. We use the directional information in velocity implicitly in the Reynolds number because similarity of flow occurs only if orientation to the flow remains the same.

   For a roughness ***Re*** (***Re***$_*$), the physical height of the roughness elements or the hydraulic roughness (or intercept parameter, z_0) is the appropriate length scale and shear velocity (not velocity at a given height, L) is the appropriate velocity scale. Both u_* and z_0 are estimated from regression of velocity versus the log of depth (***i.e.,*** from velocity at a range of heights).

   For the local ***Re*** (***Re***$_x$), L is distance from the leading edge of the flat surface and the velocity scale is the free-stream velocity.

   For a pipe or channel ***Re,*** the velocity is measured as volume per time through the pipe or channel, divided by cross-sectional area of the flow. For a pipe, it is clearly the diameter or radius that dominates flow behavior (not the length of pipe). Choices for the channel include the depth and width of the flow or their geometric mean (= square root of the cross-sectional area of the flow).

   Re encapsulates the nature of the effect of the imposed geometry on the flow, not on the object. If ***Re*** for the same object and flow geometry and orientation is the same then at a point in the flow at the same relative distance (actual distance from the object divided by L) in the same direction both the mean velocity (including the directional component of velocity) and the fluctuating component will on average be identical. If shape and orientation relative to the flow are the same for an object, then ***Re*** similarity also implies the same drag coefficient, not to be confused with the same drag force. You need more than the ***Re*** to calculate the drag force, and the ***Re*** could not predict the drag coefficient, but once you find the drag coefficient for a given shape and flow orientation, it won't change unless ***Re*** also changes. ***Re***$_*$, the roughness Reynolds number, similarly characterizes the effect of the boundary on the flow and in particular the transitions from smooth-turbulent to rough-turbulent boundary-layer structure. Imagine how much more difficult it would be if to do flow experiments if one had to vary the terms in the equation independently, one at time!

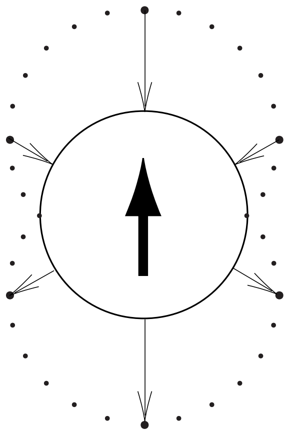
   Re$_*$ and ***Re***$_x$ both are useful for characterizing the boundary-layer flow environments in which organisms live. Pipe ***Re*** is useful in understanding flows through circulatory and respiratory systems as well as the environments of fouling organisms in pipes. Body ***Re*** is useful in characterizing the flow around organisms.

2. The first term is dynamic pressure, and is kinetic energy per unit of volume of the fluid. The second term is manometric height, also known as hydrostatic pressure. It is ordinary gravitational potential energy. The third term is static pressure, pressure produced by stalling the flow. The equation holds only along a streamline and says that the total energy in the fluid stays constant but can be converted back and forth among any of these three terms.

   A couple of you made the problem more complicated by including a boundary layer, which I specifically asked you not to include. You all threw out the irrelevant term in a horizontal flow — the middle term — because there is no change in height of the fluid. For the idealized form of Bernoulli's law given, the cylinder stops some and slows more of the upstream flow, converting kinetic energy to pressure on the upstream face of the cylinder. The first term is kinetic energy per unit of volume, which is the pressure that one could get (per unit of stopped flow) by fully stopping the flow. For both high and low ***Re,*** the pressure measured at a stagnation point corresponds pretty well to full conversion of the kinetic energy to pressure. The pressure accelerates the flow around the cylinder, which then decelerates as the obstruction is passed, returning the energy from kinetic form to pressure. So there is low pressure on both sides of the cylinder and high pressure on both the front and the rear, producing no net drag on the cylinder. In a real flow because the fluid has viscosity, some of the energy is lost to friction and is neither fully converted to kinetic energy along the sides nor fully recovered as pressure on the lee side. Energy is being lost to friction of the fluid with itself everywhere that there is a shear. If the wake is turbulent, kinetic energy is being put into fluctuating velocity components rather than a mean velocity (an unsteady flow at this scale, hence violating the assumptions), which decay into tiny vortices that pass their energy through viscosity to heat. Form drag is due to fore-aft pressure differences. They are caused at high ***Re*** by loss of kinetic energy to turbulence and viscous friction, neither of which appears in the law. Higher losses to turbulence explain why there is a higher and higher form drag (pressure difference) relative to skin friction (viscous drag on the layer of water stuck to the cylinder) as ***Re*** increases. But nevertheless both skin friction and form drag increase with increasing ***Re***. Both net skin friction and net pressure integrated over the entire cylinder point downstream. Turbulence is the big reason underlying the flat, horizontal plateau in (mean) pressure on the lee side of the cylinder in Fig. 5.2 of Vogel. Very strictly speaking, the no-slip condition means that frictional loss starts a molecular distance away from the cylinder. I introduced the idea of a tensile stress applied by the leaving fluid to the downstream side of a sphere at low ***Re***. This idea cannot apply in the presence of turbulence in the wake because inertia has overcome the tensile strength of the water, and the flow detaches. Because the flow detaches and leaves the cylinder, there is no way that the energy converted to dynamic pressure around the sides of the cylinder (low static pressure) cannot be returned to the region behind the cylinder.

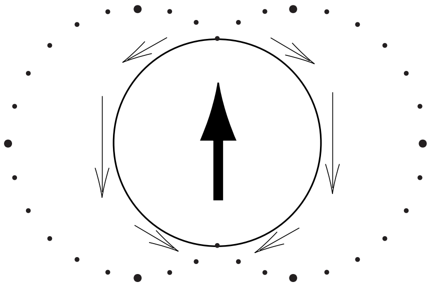
3. Most of this question is answered on pp. 158-162 in Vogel. Initially, there is no boundary layer, but a laminar one starts to form at the leading edge of all plates. The velocity gradient here is steep; shear is high. The length scale is too small initially to generate turbulence. The laminar boundary layer grows roughly as the square root of the downstream distance until it gets large enough to become turbulent. It then grows quickly for a short distance but also levels off in growth more quickly than the laminar boundary layer at the leading edge. At about the point downstream where the boundary layer becomes turbulent, a viscous sublayer (again — if you count the whole laminar boundary layer as a sublayer of itself) begins to form. Below it (in our upside-down geometry) is a turbulent boundary layer. The part of the turbulent boundary layer closest to the laminar sublayer will approximate a log layer. As most of you explained, the ultimate cause is a zero flow velocity at the plate, due to the no-slip condition. Because the big plates are longer, the boundary layer under them gets thicker. Another way to emphasize the difference is that a greater fraction of the area of a small plate will be occupied by a laminar boundary layer with high shear. So there is a viscous flow close to the plate nearly everywhere except at the laminar-turbulent transition; here I would expect sometimes to find a thin viscous sublayer and sometimes not, as symptoms of a transitional regime. Boundary layer thickness increases with x and velocity increases with z throughout. Beware of suggesting that time and distance are easily interconvertible. A fluid parcel in the viscous sublayer will travel a shorter distance in the same time as will a fluid parcel in the log layer. The drawings you produced are steady-state drawings, in which there is no change over time (steady flow) but the flow is not homogeneous in either the x or the z directions, but is in the y direction. So the picture shows change in space but not in time. In reality, the steady state would be established by the time you could make a measurement after putting in your mooring. There were two common errors. One was to draw a nonlinear plot of z vs. u and label the nonlinear part of it "laminar sublayer." In laminar flow over a plate, the velocity gradient is a straight line. The curve starts at the transitional zone between the laminar (sub)layer and the log layer. The other one was to say that because plate area does not appear explicitly in the equations involving boundary-layer growth, the plates don't differ in fluid dynamics. One plate is only 5 cm long, whereas the other is 20 cm long, and those x values enter directly into the comparison of boundary-layer thicknesses.

Pressure Distribution



Always perpendicular
to the surface in question

Shear Stress Distribution



Always tangent
to the surface in question

4. This question caused the most trouble. The setting is a small sphere moving upward slowly through an expanse of otherwise stagnant water. Nothing is making water move downward. Water is dragged upward with and by the sphere. The force doing the moving is gravity acting on the entire volume of the sphere in proportion to its excess density: $F_g = (4/3)\pi r_0^3(\rho_s - \rho)g$, but in this case the difference in density is negative, so the net force on the sphere is upward. It is really buoyancy doing the pushing; gravity is pulling both the water and sphere but pulling the water harder than the sphere, which means that hydrostatic pressure is driving the sphere upward. The map on the left is not hydrostatic pressure but rather static pressure, pressure produced by the sphere's motion being decelerated by the fluid (the third term in Bernoulli's equation earlier in your exam. The sphere's motion generates positive static pressure above it and negative static pressure below. It generates shear stresses

with the fluid, strongest near the equator of the sphere. The compression of streamlines near the sphere is very slight compared to higher ***Re***, but there is still shear due to the no-slip condition. At low ***Re*** under steady motion, shear accounts for 2/3 of the drag force on a sphere. The region of negative pressure (relative to ambient hydrostatic) on the bottom of the sphere is exactly symmetric with the positive pressure on the front but opposite in sign. Tensile strength of the fluid is literally sucking the sphere downward just as much as its compressive strength under static pressure is pushing it downward. The factor of r_0^2 in settling velocity comes from the fact that the driving force scales with r_0^3 and the retarding force, F_d, scales with r_0. When you balance one against the other, the net result is $r_0^3/r_0 = r_0^2$. If somehow you made the sphere go faster, its retarding force would increase, but its driving force cannot change, so it would slow down. Conversely, if you retarded it somehow, its drag would decrease, but its driving force would not, so it would accelerate back up to w_s. You may get confused by thinking about a body staying at constant velocity if there is no force on it versus staying at constant velocity because there is no net force on it. If you don't understand, ask questions. The body ***Re*** for this sphere is:

$$Re = \frac{\rho r_0 w_s}{\mu}. \tag{1}$$

5. τ is a shear stress, a *tangential* force *per unit of area* exerted by the moving fluid (no movement, no force per unit of area). The force is exerted on the fluid by the fluid and on any wall next to the fluid, through the no-slip condition. It has the wrong dimensions to be a force (= *ma* [M L T^{-2}]); divide a force by an area and you will get the right dimensions. It has the wrong direction to be a pressure. Dynamic viscosity is a measure of the laminar resistance to strain for a given shear rate and the proportionality factor between strain rate ($\partial u/\partial y$) and shear stress for a Newtonian fluid over distance scales or velocity gradients insufficient to turn the flow turbulent. That is, this formula would not apply well to a turbulent flow. I used *y* for the distance over which the shear acts. This formula is ***NOT*** limited to the vertical direction. By convention, *y* is the cross-stream direction. A clearly relevant geometry that follows convention would be flow past a flat plate perpendicular to the flume bottom and parallel to the flume axis. If you know viscosity, and know the gradient in velocity just next to an object and the object's area, you can calculate the shear force on the object. Several people tried to apply the equation for viscous shear stress to very large scales. That won't work because the relevant Re gets huge from the large spatial dimension [L] involved, making the flow turbulent. The equation does not apply to turbulent shear. It applies to a viscous sublayer or a laminar layer (*e.g.,* on the leading edge of a plate before it grows large enough to become turbulent).