MAE 571, Fall 2014, Homework #5
Due Wednesday, November 19

Prob 1. (50%) Use Ansys-Fluent to simulate the flow in a simple circular pipe, as illustrated in Fig. 1. The pipe has L and D as its length and diameter. A uniform inflow velocity, U, is imposed at the left opening (the inlet). The right opening of the pipe is treated as an open outlet. The purpose of this exercise is to use numerical simulation to study the behavior of the boundary layer. Please make sure that a sufficient mesh resolution is used near the boundary (see Tutorial for further detail). For the convenience of defining the tasks, we will take the axis of the pipe (the red line) as the x-axis, with the y-axis points upward as shown in Fig. 1.

(a) As a starting point, put water in the pipe and set $U = 1$ cm/s, $L = 50$ cm, and $D = 10$ cm. Set "Model" to "Viscous, Laminar" and seek a steady solution. Make a contour (or color-fill) plot of the u-velocity in the rectangular domain, $ABCD$, which is the intersection of the x-y plane and the interior of the pipe (see Fig. 1). Make a plot of selected profiles of u (in the $ABCD$ plane) as a function of y at different values of x. If the basic set-up described above does not produce a clear boundary layer, try to adjust it until one is produced. Use that case instead to make the required plots. Describe how the boundary layer thickness varies with x (i.e., the distance from the inlet).

(b) Using the basic case in (a) as the reference, vary U and the kinematic viscosity of the fluid, ν, to examine how the structure of the boundary layer changes with those two parameters. For example, in the Blasius solution, qualitatively the boundary layer would become thicker with an increasing ν but thinner with an increasing U. Would that be the case in this system? (Note that the Blasius solution is not applicable to this system with a finite, enclosed domain. We bring this point up only to stimulate the discussion. This exercise is not intended as a verification of the Blasius solution in any way.)

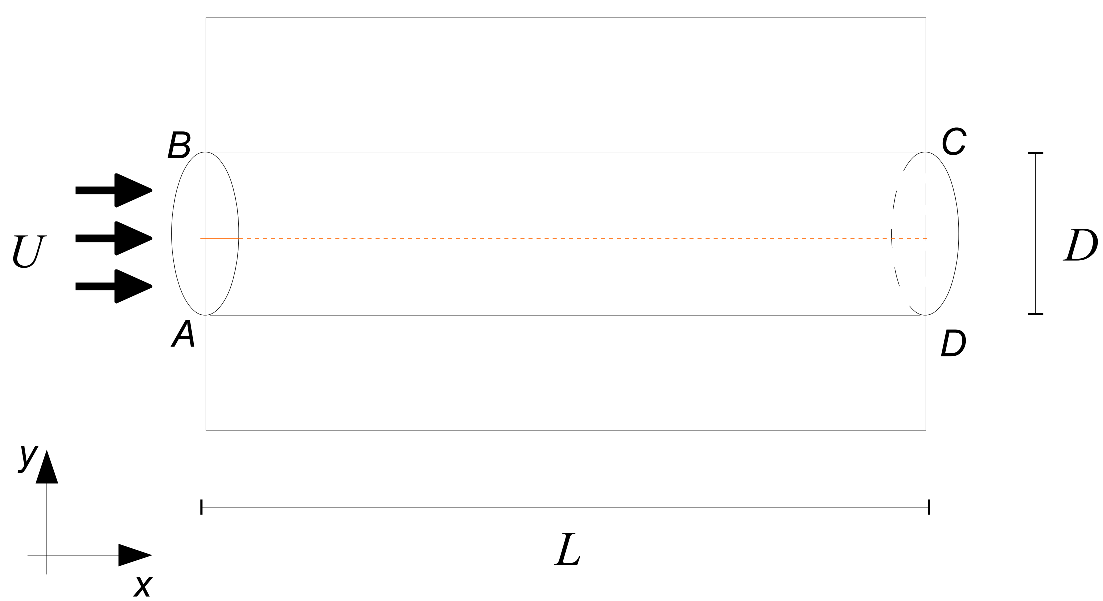

Fig. 1

Prob 2. (50%) As discussed in class, boundary-layer separation may happen in the presence of an adverse pressure gradient (i.e., with a pressure gradient force that counters the flow). Therefore, for a steady incompressible flow in a pipe with non-uniform diameter, separation is more likely to happen in a "diverging" segment (with an increasing diameter of the pipe down stream) than a "converging" segment. Use Ansys-Fluent to simulate the flow in two prototypical pipes, one with a bloated mid-section and the other with a bottleneck (cases A and B in Fig. 2, not drawn to scale), to illustrate that separation indeed happens only over the diverging segments (indicated by red bars in Fig. 2). It is up to you to design the detail of the pipes and the flow systems, but a suggested starting design for a pipe filled with water would be with $U \sim 1$ cm/s, $L \sim 1$ m, $D_{\max} \sim 20$ cm, and $D_{\min} \sim 10$ cm. (If the suggested setting does not work, please feel free to adjust it.) Again, use the parameter setting of "Viscous, Laminar" and "Steady solution" in Ansys-Fluent. You may use the contour or color-fill plots of *u*-velocity in one of the cross sections (in the same fashion as Prob 1.) to display the key outcome of the simulation. One plot for each of case (A) and (B) will suffice, but please feel free to show more cases if they serve to address the robustness of the results. Due to the axial symmetry of the system, the separation of the flow may occur randomly at a particular sector of the cylinder. In that case, it is useful to try different cross sections to get the best picture of the relevant structure of the flow.

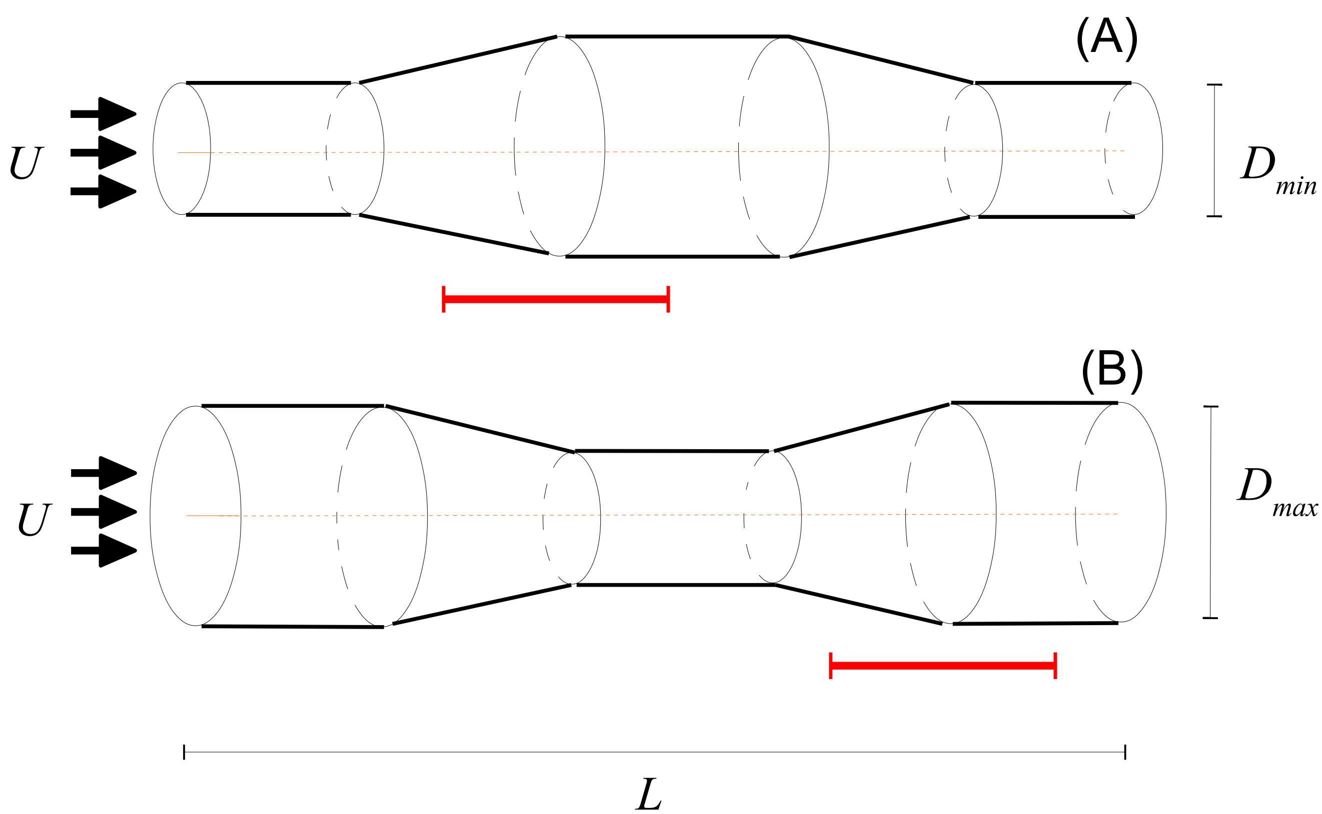

Fig. 2