MAE 571, Fall 2016, Homework #3 (12 points)

Prob 1. (6 points) In this exercise, we 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 velocity inlet). The right opening of the pipe is set up as a pressure outlet. We assume that the pipe is filled with water of constant density and viscosity. "Energy equation" can be turned off in Fluent. We aim to examine the behavior of the boundary layer by numerical simulation. It is recommended that an enhanced mesh resolution be used near the boundary - see Tutorial #1 for detail. To define the tasks more precisely, we will take the axis of the pipe (the red line) as the x-axis, with the y-axis pointing upward as shown in Fig. 1.

(a) As a starting point, set $U = 0.5$ cm/s, $L = 50$ cm, and $D = 10$ cm. Set "Model" to "Viscous, Laminar" and seek the steady solution. Make a contour (or color-fill) plot of the u-velocity in the rectangular cross section, $ABCD$, 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 picture of boundary layer, try to adjust it until one is produced. Use that case instead to make the required plots. Describe how boundary layer thickness varies with x (i.e., the distance from the inlet). [At low Reynolds number, if the pipe is long enough the "boundary layer" from all sectors eventually merge and the flow approaches a parabolic profile. We are more interested in the behavior of the boundary layer before this complete merger occurs.]

(b) Using the basic case in (a) as the reference, vary U and the kinematic viscosity of the fluid, ν, to examine how the thickness of boundary layer changes with those two parameters. For example, in the Blasius solution (to be discussed in class), boundary layer would become thicker with an increasing ν but thinner with an increasing U. Would that be the case in this system? (Our system is different from the idealized "flow over a flat plate" that leads to the Blasius solution. This exercise is not intended as a verification of the Blasius solution in any way.)

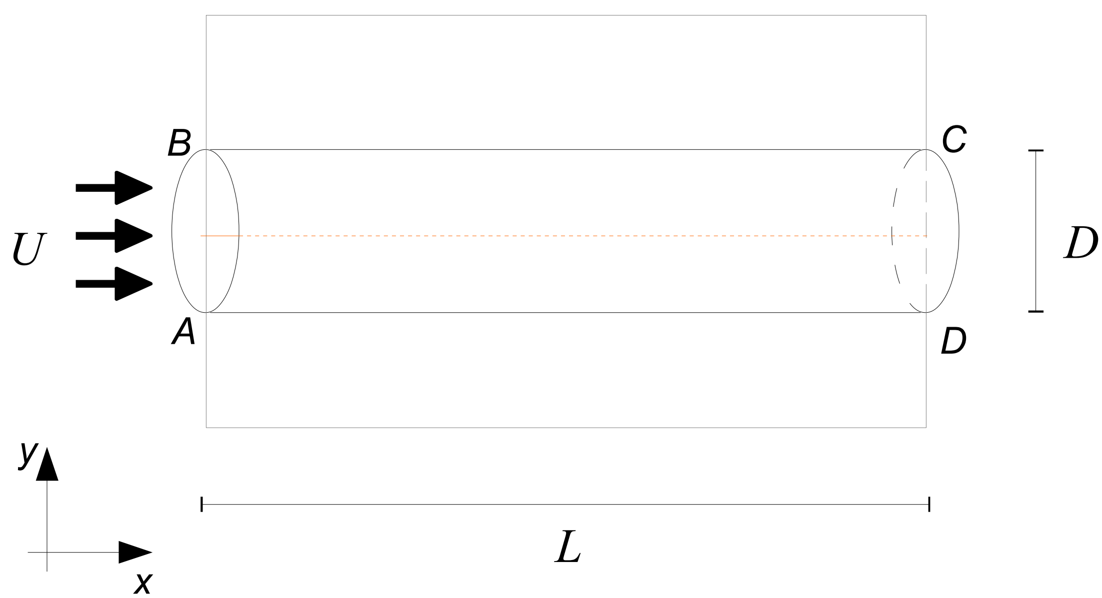

Fig. 1

Prob 2. (6 points) As discussed in class, flow separation in boundary layer may occur under the influence 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 occur in a "diverging" segment (with an increasing diameter of the pipe down stream) than a "converging" segment. This exercise will use Ansys-Fluent to simulate this phenomenon.

Consider the flow in two prototypical pipes, one with a bloated mid-section and the other with a bottleneck (Case A and B in Fig. 2, not drawn to scale). Perform Ansys-Fluent simulations for the two flow systems to demonstrate that flow separation indeed occurs 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, $L \sim 50$ cm, $D_{\text{max}} \sim 10$ cm, and $D_{\text{min}} \sim 5$ cm. (If the suggested setting does not work well, 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 plot 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 cases (A) and (B) will suffice. However, please feel free to show the results from 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 obtain the best picture of flow separation.

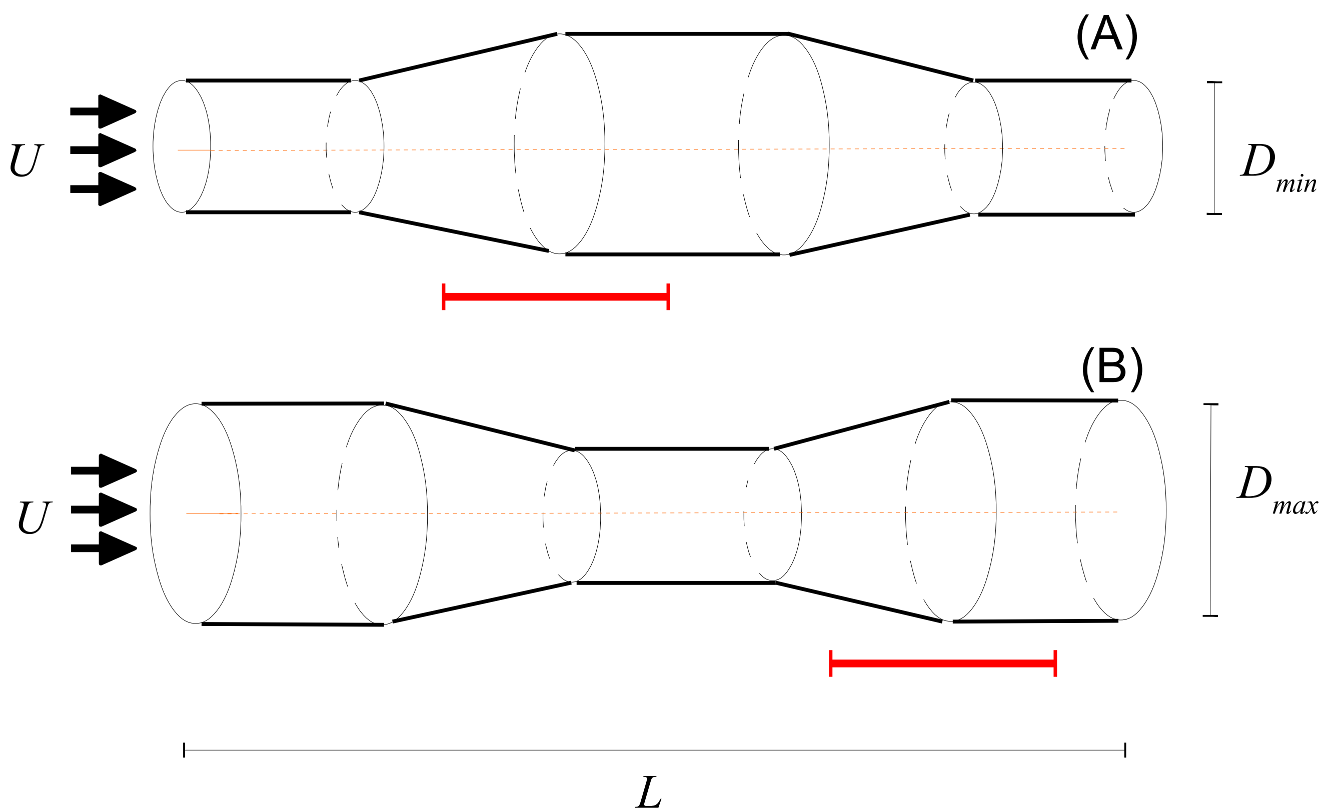

Fig. 2