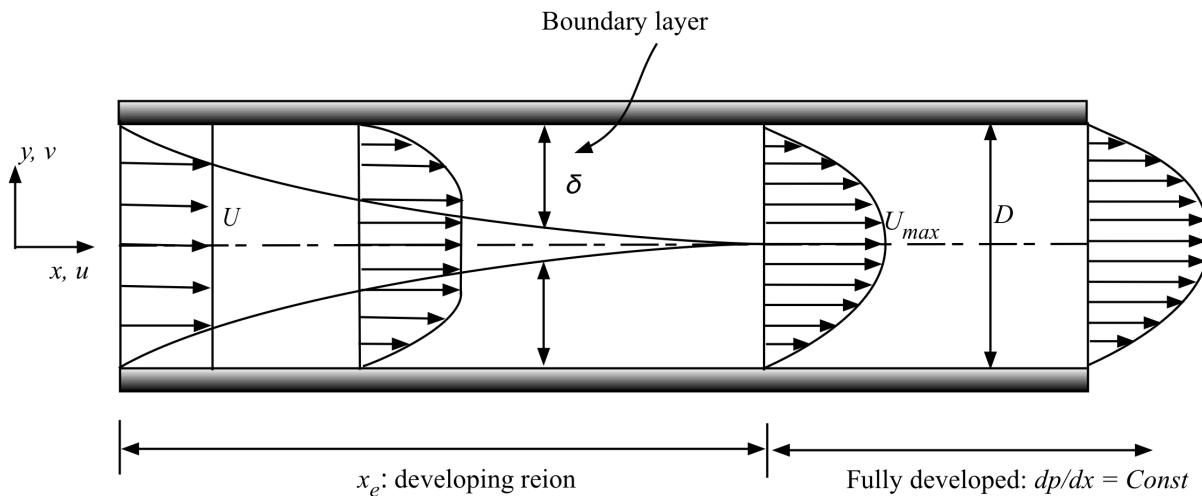


CFD Project (MEMS 1071, Applied Fluid Mechanics)

(Due by the class on Feb 27, 2017)

You are doing numerical simulations on incompressible, 2-D, viscous laminar channel flow as shown in Figure below using ANSYS (Fluent). There are two distinct regions in the channel flow: developing region and fully developed region. In the developing region, two boundary layers grow from the channel leading edge and eventually merge together at the center of the channel far downstream. In this region, there does not exist an exact solution. The entrance length x_e denotes the streamwise distance from the leading edge to the position where the boundary layers meet. Once the flow passes the developing region, it reaches the fully developed region where the velocity profile remains the same regardless of the streamwise position. In addition, the pressure gradient (dp/dx) also remains constant. You can easily find the exact solution of the velocity profile for this region in the textbook (parabolic profile). The goal of this project is to numerically find the relation of x_e/D to Re_D by addressing the sub questions below.



(a) Repeat the simulation for $Re_D = 50$ we did in the class. **For your practice, however, you need to select different dimensions and inlet velocity from those demonstrated by the graduate student in the class.** Plot the axial velocity u/U vs. y/D in 10 different axial positions including the developing and fully developed regions. To compare your results with the exact solution, you need to plot them together with the exact solution.

(b) Using the numerical results above, find dp/dx and Q (volume flow rate) in the fully developed region. Check their relation against the exact solution ($\frac{Q}{w} = \frac{-D^3}{12\mu} \frac{dp}{dx}$ where w is the channel width).

How much error do you have?

(c) Find the entrance length x_e/D for the above numerical results. In order to find a relation of x_e/D to Re_D , you need to carry out simulations for 7 different Reynolds numbers in the range of $Re_D < 1000$. Compare your result with that in literature.

*** One of the main purposes of this project is that everybody runs Fluent and has an experience on CFD. To prove this, everybody must attach screenshots of mesh generations (overview and zoom-in view near the walls) and converging histories for each CFD simulation in the appendix. Or you would not have any grade on this project ***