

Case (a), (b), and (c1) : We have discussed in class about the dependence of the solution on the detailed parameter setting in ANSYS-Fluent. We have also discussed the reason why the calculation by the steady-state solver in ANSYS-Fluent may not converge for the cases with a sufficiently large inlet velocity. In short, those cases are in the high Reynolds number regime with the actual (time-dependent) solution being either a stable oscillation or a turbulent flow. A steady state, if it exists, would be unstable. (Note that even with the inlet velocity set to 5 cm/s the Reynolds number of the system is already on the order of 10000.) In those cases, mesh refinement would not necessarily improve convergence or lead to a better solution. One of the purposes of this exercise is for you to see the subtlety of this situation.

Case (c2) : In this case, switching the "model" setting to "k-epsilon" would allow ANSYS-Fluent to treat turbulence. More precisely, the effect of turbulence on the mean flow is calculated. The "convergence" in this case means the convergence of the solution for the mean flow (and the effect of turbulence reaches a statistical equilibrium - not a fixed steady state). With a sufficiently high mesh resolution, you may find the solution to approach a typical fully developed turbulence state with a near-flat velocity profile (for the mean flow) except within the very thin boundary layers near the walls. (See the last example at the end of this document.) In this state, the u-velocity along the x-axis should be approximately inversely proportional to the ratio of the local cross-sectional area to the cross-sectional area at the inlet, due to incompressibility and the conservation of mass. If the diameter of the pipe as a function of  $x$  is  $D(x)$ , the diameter at the inlet is  $D_{inlet}$ , and the u-velocity at the inlet is  $u_{inlet}$ , then the u-velocity along the x-axis would be approximately

$$u(x) = u_{inlet} \left( \frac{D_{inlet}}{D(x)} \right)^2 ,$$

as illustrated in Fig. 1. The solution for case (c2) with a very-high resolution mesh and "k-epsilon" option is close to this ideal state.

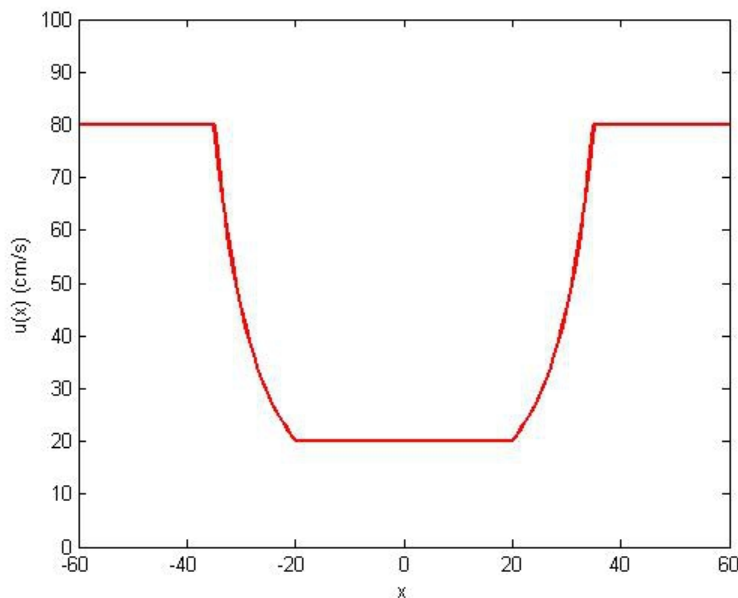
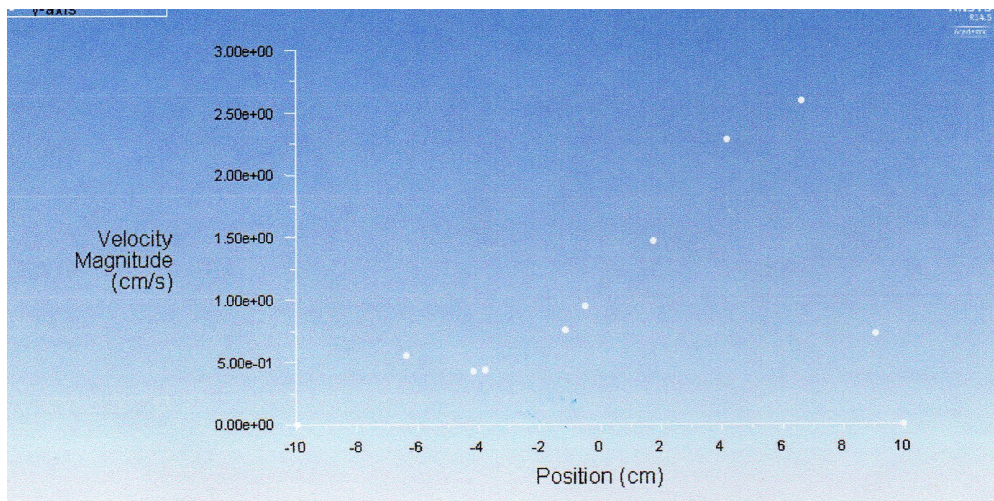
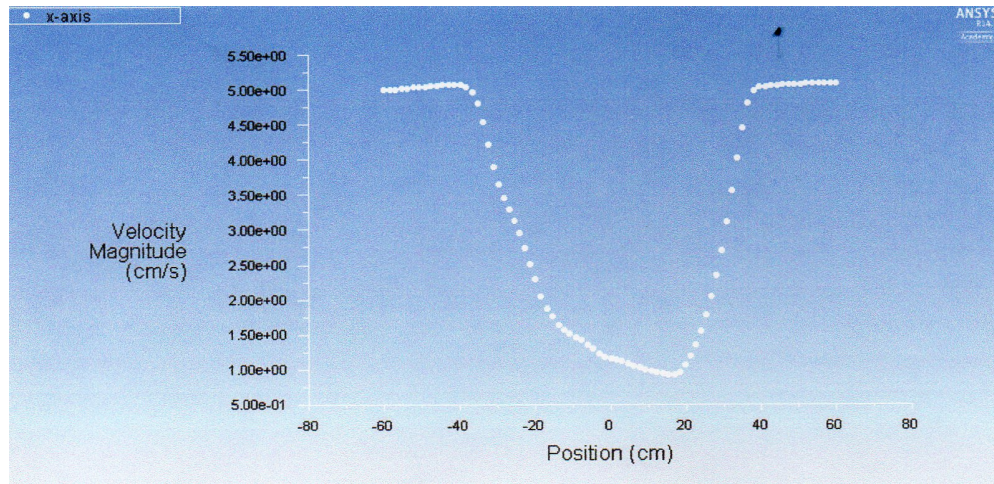
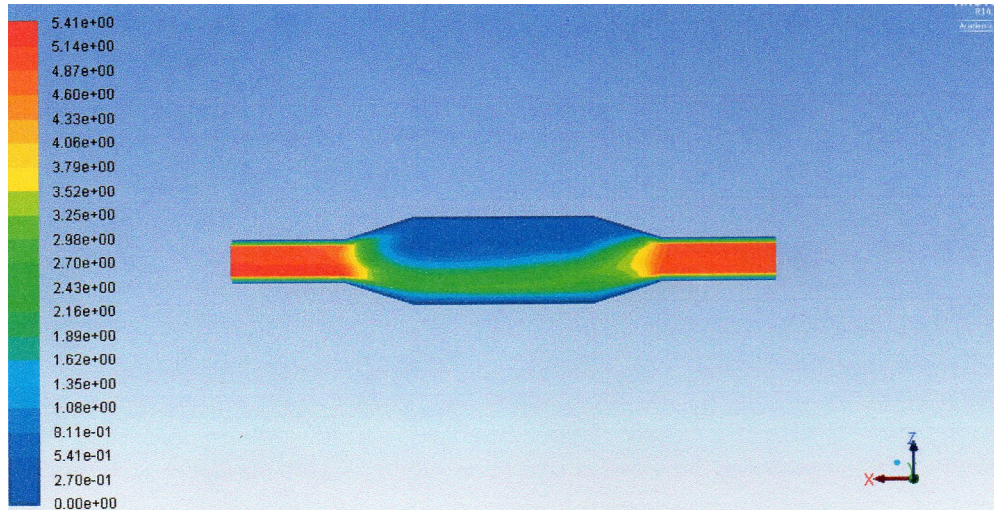


Fig. 1

There are no definite answers to the questions in this homework. The following are but a few selected examples of the solutions.

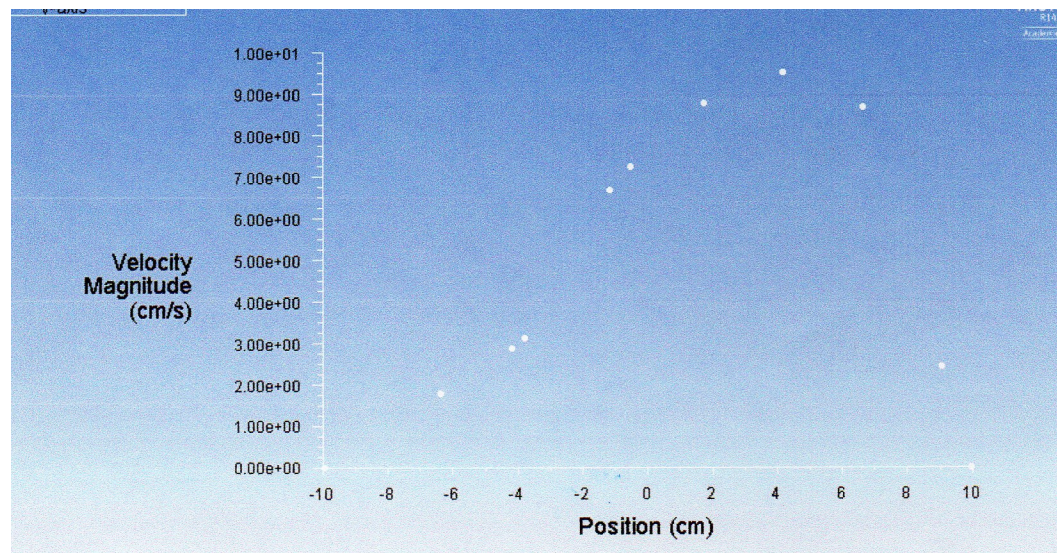
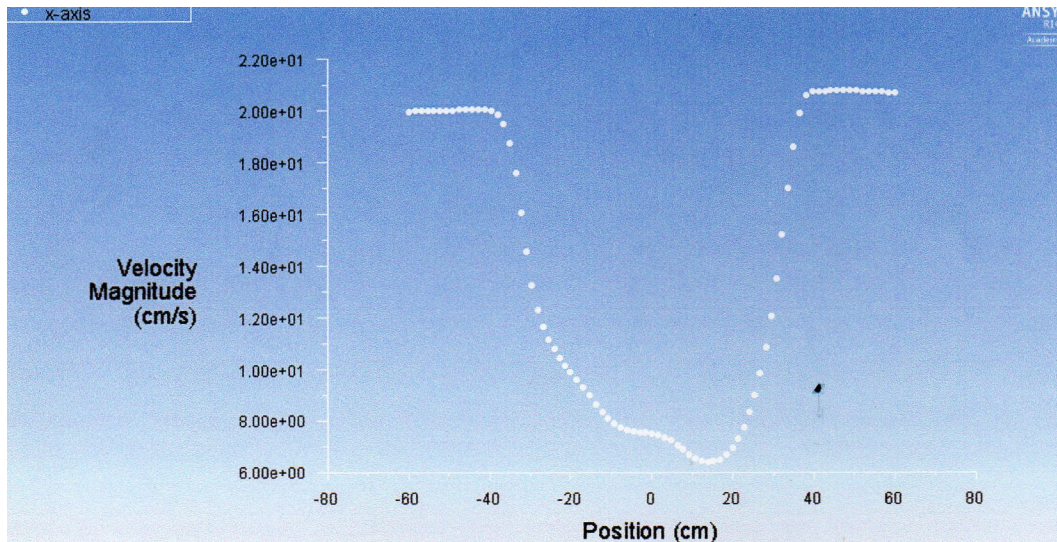
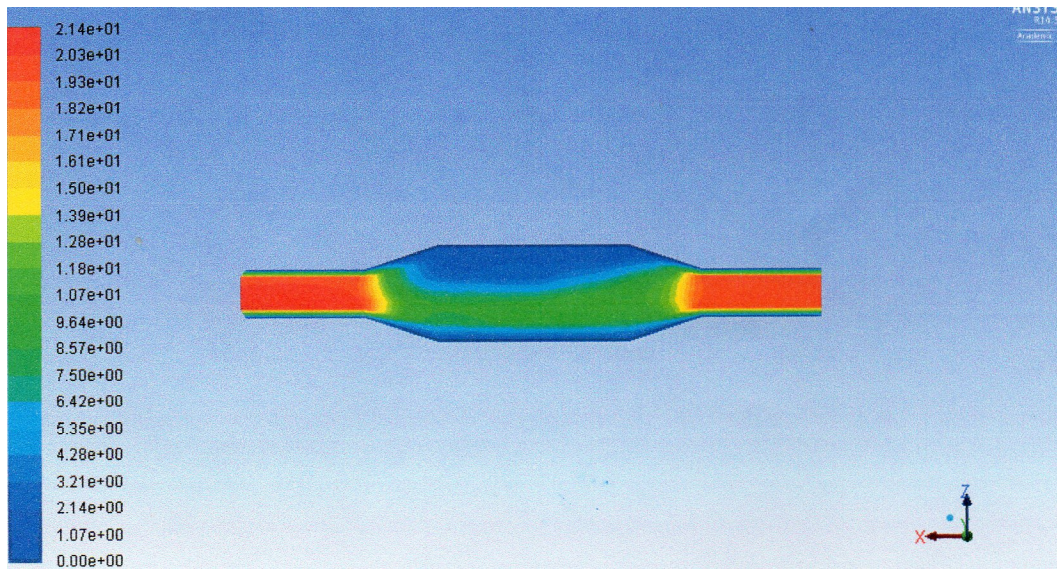
### I. ANSYS-Fluent with default mesh (4374 nodes) (Thanks to Aidan Coyle)

(a)  $u_{inlet} = 5 \text{ cm/s}$ , Model set to "Laminar"



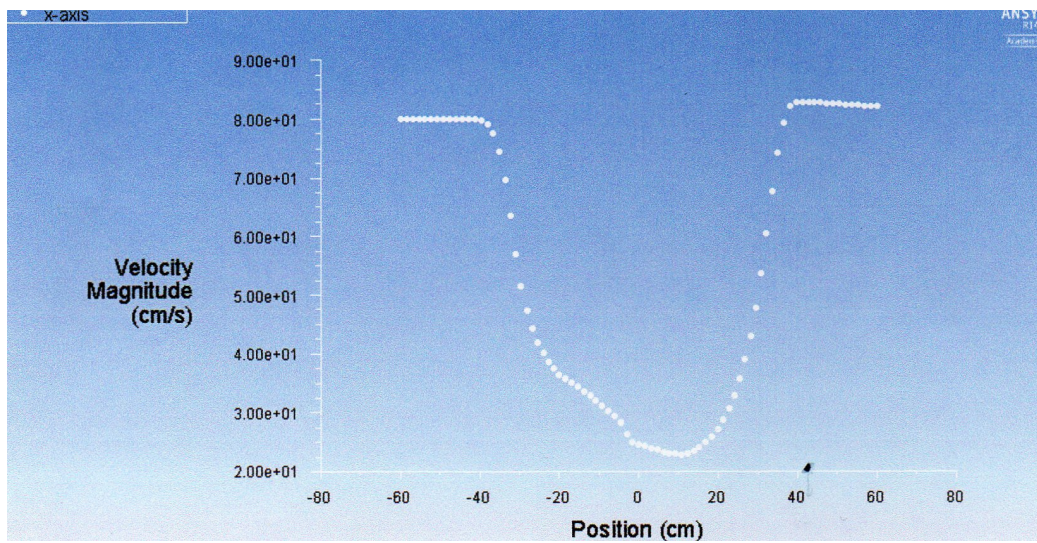
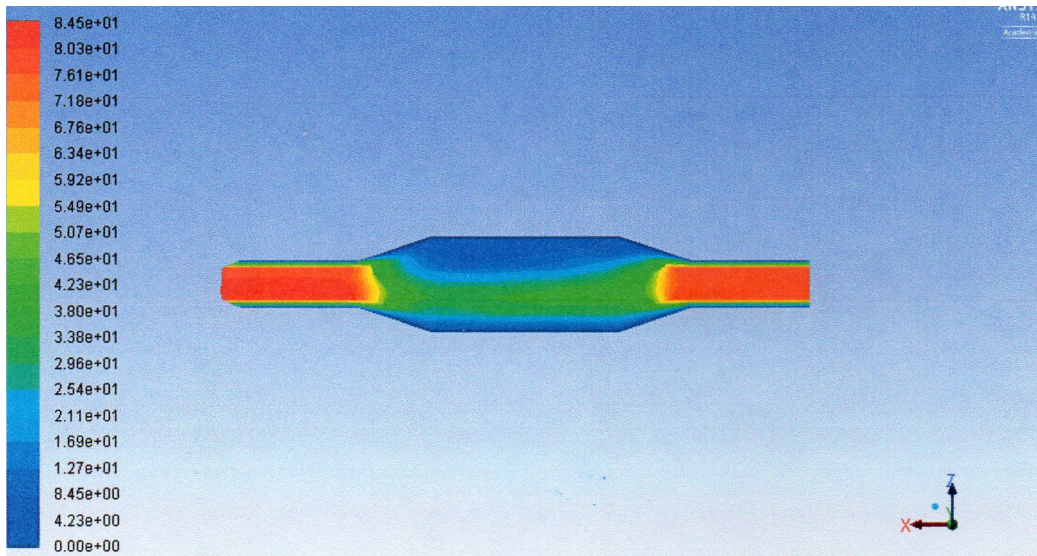


(b)  $u_{inlet} = 20 \text{ cm/s}$ , Model set to "Laminar"



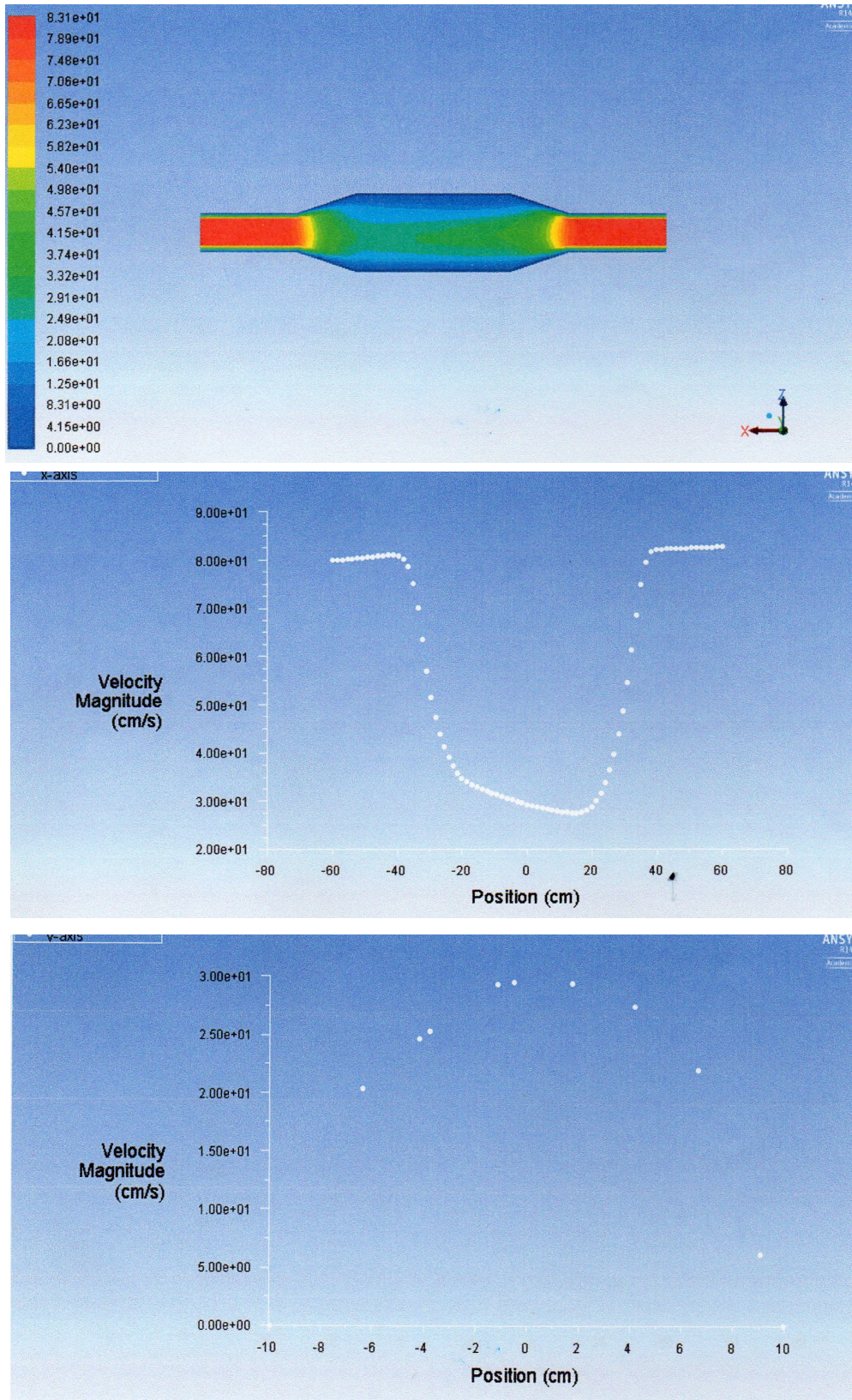


(c1)  $u_{inlet} = 80 \text{ cm/s}$ , Model set to "Laminar"





(c2)  $u_{inlet} = 80 \text{ cm/s}$ , Model set to "k-epsilon"

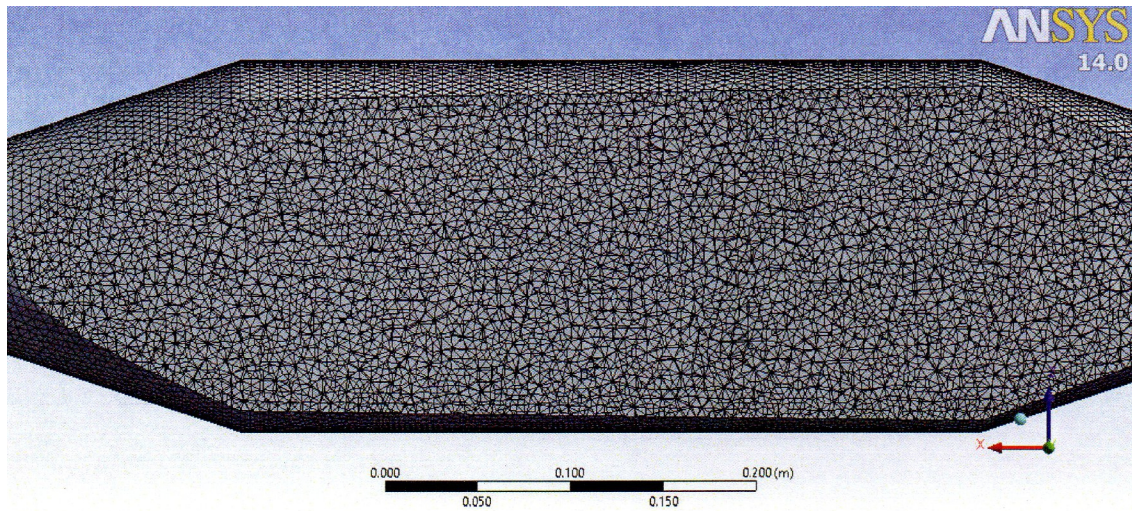


(Note that with a low resolution the u-velocity along the y-axis looks more like a parabolic profile.)

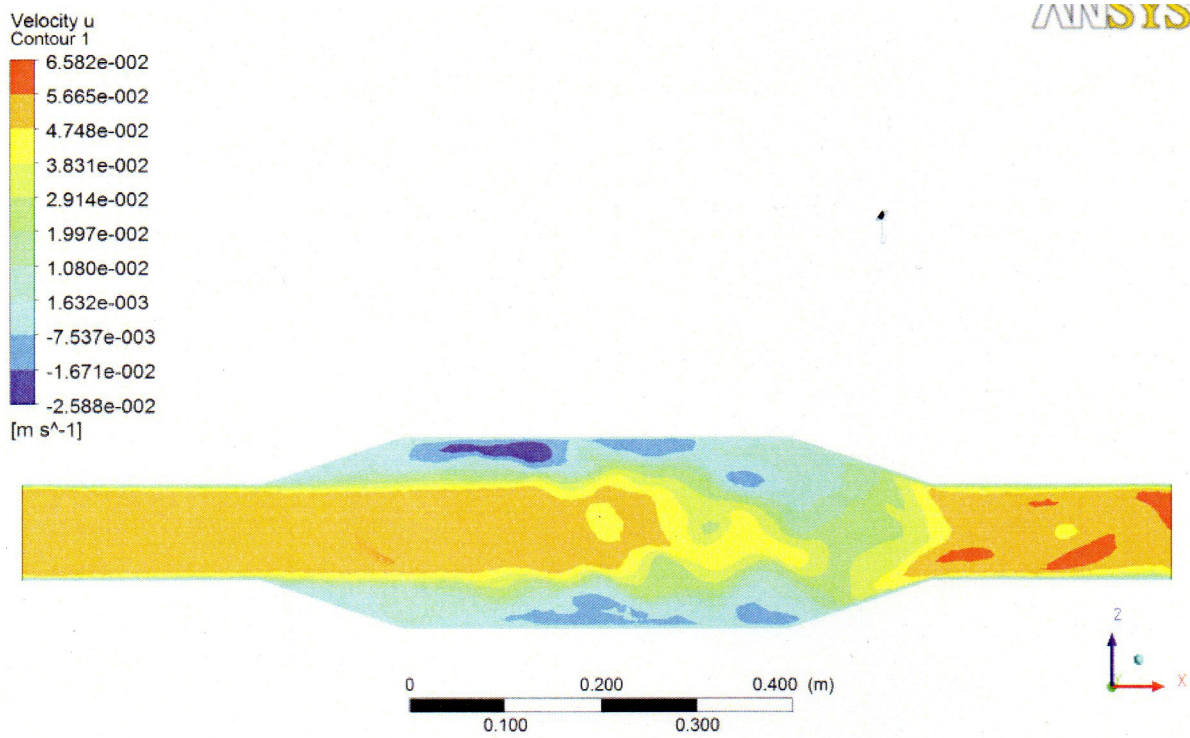


## II. ANSYS-Fluent with refined mesh (116975 nodes) (Thanks to Aditya Bhagwat)

The refined mesh:

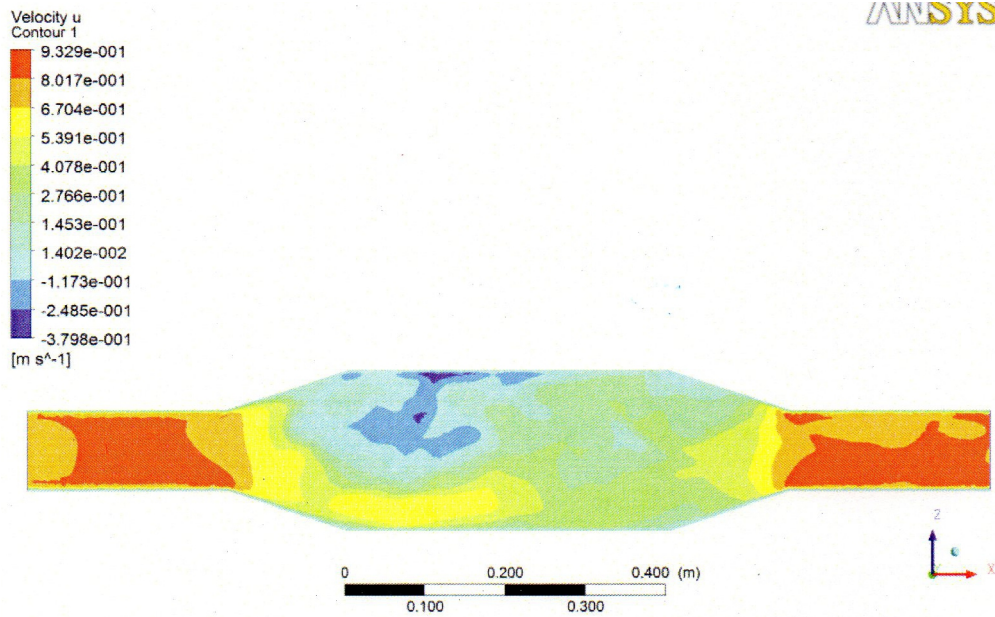


(a)  $u_{inlet} = 5 \text{ cm/s}$ , model set to "Laminar"

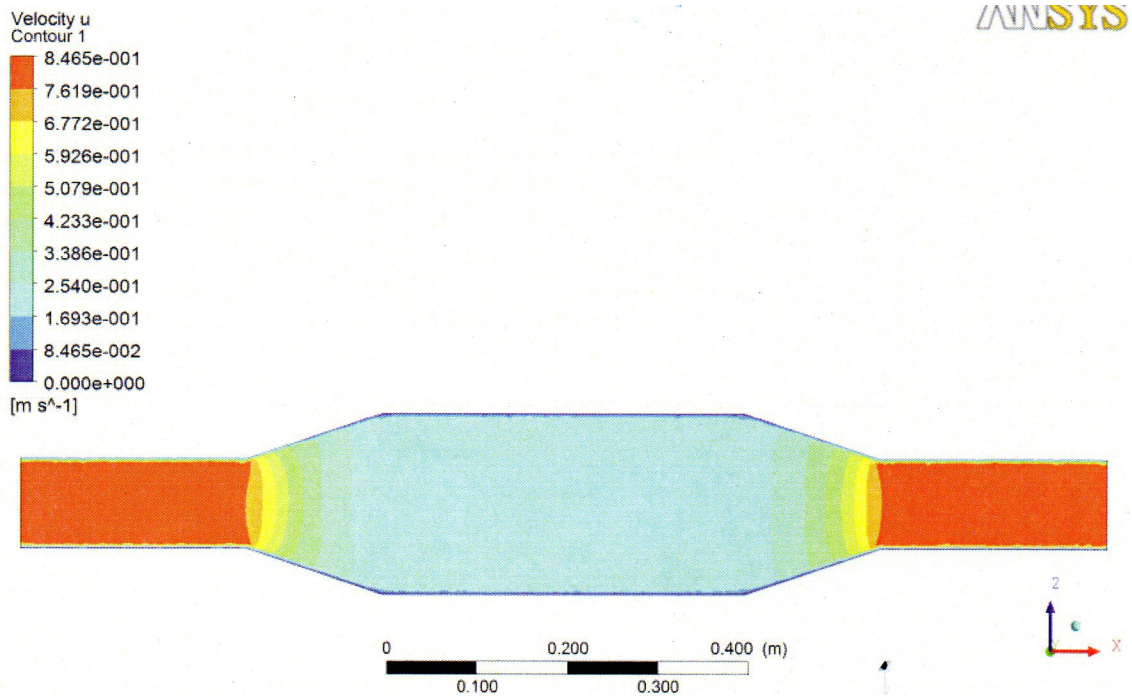




(c1)  $u_{inlet} = 80 \text{ cm/s}$ , Model set to "Laminar"



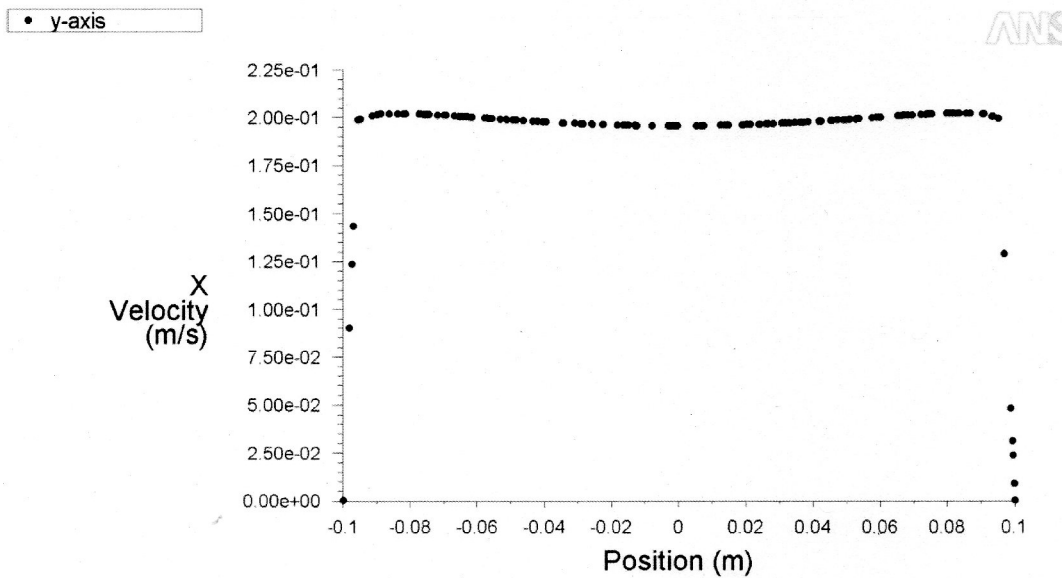
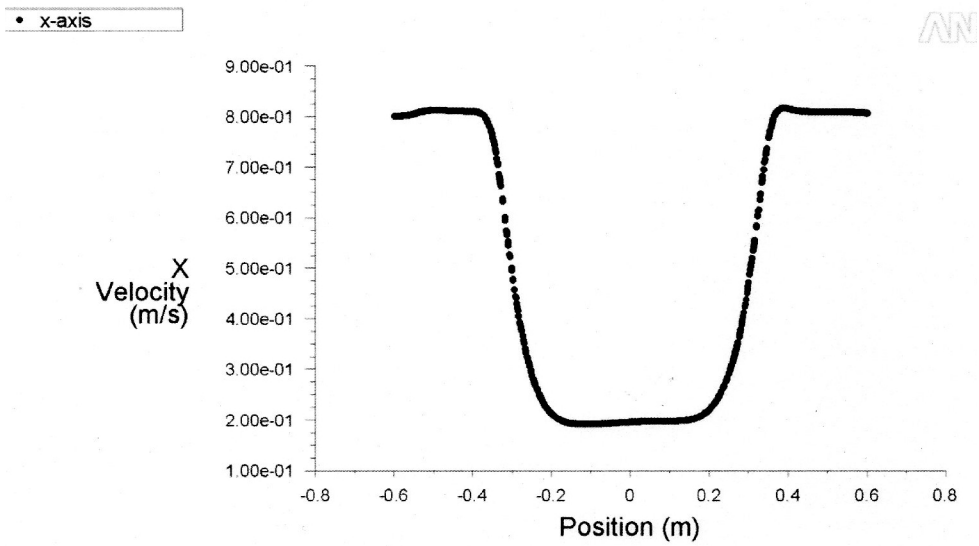
(c2)  $u_{inlet} = 80 \text{ cm/s}$ , Model set to "k-epsilon"





(continued)

(c2) Profiles of u-velocity along the x-axis and y-axis



See instructor's note in p. 1 for relevant discussions on these two plots.

Final note: For the simulation of a turbulent flow, the desirable mesh/grid resolution usually increases with Reynolds number.