MAE561/471 Fall 2013 HW2

This exercise is for you to get familiar with the end-to-end setup for fluid dynamical simulations using ANSYS-Fluent. A step-by-step tutorial (hereafter called the Tutorial) will be provided by the instructor separately.

No restriction on discussion with peers for this exercise. You are encouraged to share your experience on ANSYS-Fluent with fellow students.

1. Work through the <u>entire Tutorial</u> step by step to ensure that you are able to set up the geometry, mesh, and fluid dynamical parameters, then execute the simulation for the main example of a flow in a system of pipes.
(a) After completing Step 5 in the Tutorial, make screenshots of the contour/color maps of velocity and temperature at the plane of symmetry (essentially Figs 1.21 and 1.22 in the Tutorial) and include them in your report.
(b) Repeat Steps 4-5 but artificially triple the viscosity of water from 0.0008 to 0.0024 (Pa)(s) and re-run the simulation. Include the screenshots of the contour/color map of velocity and temperature (in the same format as Figs. 1.21-1.22) in the report. See p. 31 of the Tutorial on how to change the fluid parameters.
(c) Work through Steps 6-9 but consider a minor variation: In Step 7, action 2(b) (in p. 54), instead of changing the "R1" of the small inlet from 0.5 to 0.75 in, we will change it to 1 in (i.e., double the diameter of the inlet from 1 to 2 in.). Work through the rest of the steps and save (i) a screenshot of the updated mesh (equivalent to Fig. 1.33) and (ii) the contour/color plots of velocity and temperature (in the same format as Figs. 1.21-1.22) for the report. For Part (c), please use the original value of viscosity (0.0008) for water.

Note: Please ignore the instruction in p. 4 to "download elbow-workbench.zip ...". That is for the case when we choose to skip the first step of creating the geometry file. In this exercise, we do want to create the geometry file from scratch.