MAE 494/598 Homework 1 (5 points)
Note: 1 point ≈ 1% of the total score for the semester. Hard copy of report is due *at the start of class* on the due date. No electronic submission. No restriction on collaboration for this assignment. *If you receive help from other student(s), it must be acknowledged in the report.*

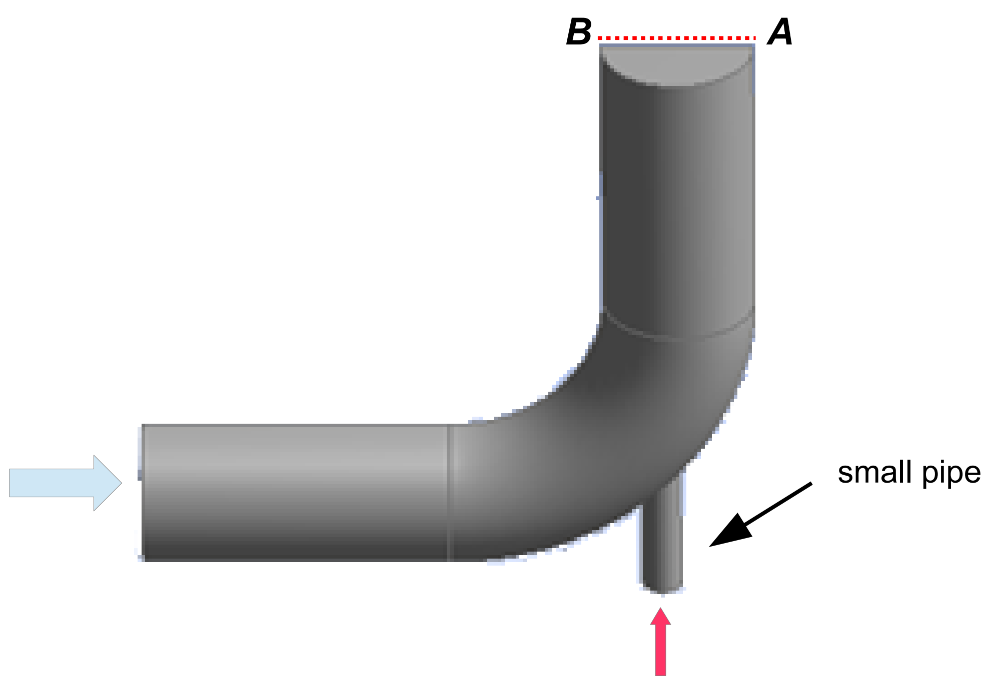
Task 0 (for self exercise only; no need to show results)
Complete Tutorial #1 and #2 for Ansys-Fluent.

Task 1
Repeat the task in Tutorial #1 but with the following modifications:

(i) Follow Step 7 (p. 1-54) in Tutorial #1 to enlarge the "small pipe". Instead of *0.75 in* as used in the tutorial, increase the radius of the pipe to $R = 1.0$ *in*.
(ii) Reduce the inlet velocity associated with the "small pipe" to $V_{inlet} = 0.6$ m/s.

Otherwise, follow the remaining steps to complete the simulation. Under the new setting, produce the contour plots of velocity and temperature in the plane of symmetry (i.e., the counterparts of the right panels of Figs. 1.41 and 1.42). In addition, make a contour plot of temperature over the circular opening of the outlet.
(Tip: To ensure a smooth connection between the small pipe and the main pipe, when the radius of the small pipe is increased to *1.0 in* you need to also increase the "depth" of extrusion as described in point "x." in p. 1-15 of the tutorial.)

Task 2
Using the simulation from Task 1, make the line plots of temperature and velocity (normal to the outlet) along the line, *AB*, as indicated in the diagram below. This is the line on the plane of symmetry that passes through the center of the outlet. Compare those profiles with their counterparts from the standard case in Task 0 (i.e., the case with $R = 0.5$ *in* and $V_{inlet} = 1.2$ m/s) Please make two separate plots for the profiles of velocity and temperature. Each plot should show two curves, one for the standard case from Task 0 and one for the modified case from Task 1.



Task 3

In this exercise, we will check the *energy balance* and *mass balance* of the system. All the calculations in this task will consider the simulation from Task 1. The flow rate of heat through a given surface, A, is defined by

$$H \equiv \iint_A v_n \, \rho \, C_p \, T \, dA \quad , \qquad \text{Eq. (1)}$$

where ρ and C_p are density and specific heat of water, v_n is the velocity normal to the surface, and T is temperature. (Thus, H has the unit of power, in J s^{-1}.) The flow rate of mass through a given surface, A, is defined by

$$M \equiv \iint_A v_n \, \rho \, dA \quad . \qquad \text{Eq. (2)}$$

(Note that H is the area integral of heat flux while M is the area integral of mass flux.) Since there is no internal heat source or sink in this system, in the steady state the H for the outlet should equal the sum of the H's for the two inlets. Likewise, since the system has no internal mass source/sink, the M for the outlet should equal the sum of the M's for the two inlets. Calculate the three values of H and three values of M to show that the requirements of energy and mass balance are satisfied.

Hint: Be aware that while T and v_n are constant at the inlets, they are non-uniform at the outlet. Fluent has built-in functions to perform the integration in Eq. (1) or (2) when T and/or v_n are non-uniform across the surface, A. It is part of your job to discover those relevant functions. Note that Fluent adopts the convention that mass (or heat) flow *into the system* and *out of the system* have opposite signs. Alternatively, the integration in Eq. (1) or (2) could be performed by first dumping the data of T and v_n to a text file, then using other tools (e.g., Matlab) to complete the integration. This approach might be slightly cumbersome because the velocity and temperature are generally given at irregularly spaced nodal points. In your solution, please briefly describe the method you choose to evaluate H and M in Eqs. (1) and (2).