MAE 494/598 Applied CFD, Fall 2017   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) to complete this assignment, it should be acknowledged in the report.*

Task 0 (for self exercise only; no point and no need to show results)
Complete Tutorial #1 and #2 for Ansys-Fluent. At the minimum, you should be able to reproduce the plots of velocity magnitude & temperature fields in Figs. 1.21 & 1.22 in Tutorial #1, and velocity vector field in Fig. 1.7 of Tutorial #2. (Minor differences between your results and those figures are acceptable. We do not expect perfect replication for a turbulent flow.)

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 <u>radius</u> of the pipe to $R = 1.0 \ in$ (i.e., increase the <u>diameter</u> of the pipe to *2.0 in*).
(ii) Reduce the inlet velocity associated to the "big pipe" to $U_x = 0.01$ m/s"; Reduce the inlet velocity associated to the "small pipe" to $U_y = 0.03$ m/s.
(iii) Change the setting of "model" from "k-epsilon" to "laminar". When the model is set to laminar, there is no longer the need to set "turbulence intensity" in the boundary conditions for the two inlets.

Otherwise, follow the remaining steps to complete the simulation. From this new simulation, produce the contour plots of the <u>velocity magnitude</u> and <u>temperature</u> in the plane of symmetry, in the format similar to Figs. 1.21 & 1.22 in Tutorial #1. In addition, produce a plot of the velocity vectors in the plane of symmetry in the format similar to Fig. 1.7 of Tutorial #2.

Tips:

(i) 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* one needs to also <u>increase the "depth" of extrusion</u> in step "x." in p. 1-15 of Tutorial #1. With the enlarged "small pipe", the value of Hydrolic Diameter in p. 1-37 of Tutorial #1 should be changed from *1.0 in* to *2.0 in*.

(ii) The quality of contour plots as shown in Figs 1.21 & 1.22 in Tutorial #1 can be improved by using CFD-Post or adjusting the options under Results. Examples of refined plots are shown in Figs. 1.25 & 1.26 in Tutorial #1. A notable option, useful for saving ink for the report, is to replace filled contours by unfilled contours, i.e., just contour lines with no color fill in between. Similarly, there are options to reduce the density of vectors for plotting the velocity vector field.

Task 2
From the simulation in Task 1, produce the profiles (line plots) of temperature and velocity (the component normal to the outlet) along the line, *BA*, as indicated in the diagram below. This is the line on the plane of symmetry that passes through the center of the outlet. Please make two separate plots for the profiles of velocity and temperature.
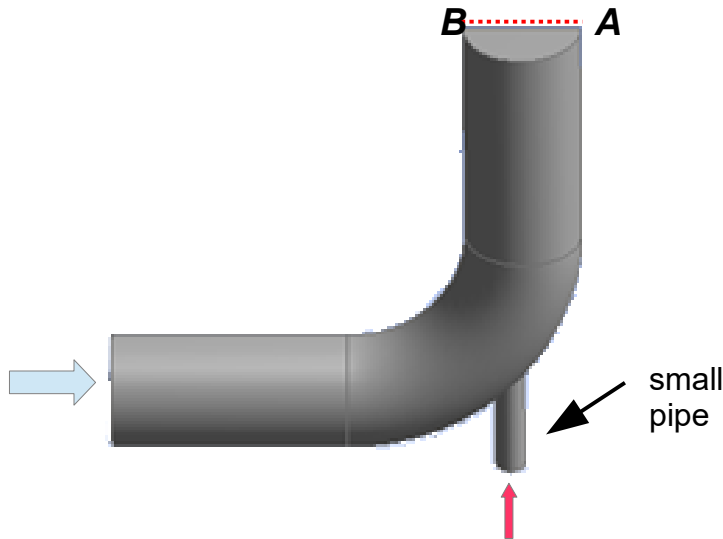
Tips: (i) To produce a line plot along a particular line, one can first manually define that line as a *New Surface* in the window for x-y plot. To make this work, one needs to know the coordinates of points A and B. (The "new surface" is defined as the line connecting the two points.) It is useful to take note of this particular information in the beginning when one creates the geometry.   (ii) The line plot can be improved by first "dumping" the data to a text file, then using Matlab to do the plotting. The data file can be created by selecting "write to file" in the window for x-y plot under Results.

Task 3

Calculate the total rates of heat transfer, H, across the two inlets and one outlet. Here, the quantity we are interested in is defined by

$$H \equiv \iint_{A} v_n \, \rho \, c \, T \, dA \ , \hspace{4cm} \text{Eq. (1)}$$

where ρ and c are density and specific heat of water, v_n is the velocity normal to the surface of the opening of the inlet or outlet, and T is temperature. The integral is carried out over the surface of the inlet or outlet. Thus, H has the unit of power, in J s^{-1}. Be aware that v_n and T are generally not uniform across the opening of the inlet/outlet. In other words, the integral in Eq. (1) cannot be simply replaced by "$v_n \rho c T A$" where A is the area of the cross section and v_n and T are some averaged constants. The deliverables for this task are the three values of H corresponding to the inlets and outlet, and a brief description of the procedure used to compute those values. Since there is no internal heat source or sink in this system and the effect of thermal expansion (which facilitates the conversion between thermodynamic and kinetic energy) is turned off by setting density to constant, by conservation of thermodynamic energy we expect the sum of the values of H for the two inlets to equal the value of H for the outlet.

Tips: (i) To combine the product of two or multiple fields into a single field, try to use "Custom Field Function" as explained in p. 1-36 in Tutorial #2. For example, one can first define a new variable, $Z \equiv v_n \rho c T$, then use a separate function to perform the surface integral of Z in order to compute H.

(ii) Fluent has built-in functions for performing surface or volume integral for a given variable. Those functions are sufficient for producing the deliverables for this task all within Fluent.