MAE 494/598 Applied CFD, Fall 2018   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, velocity vector field in Fig. 1.7 of Tutorial #2, and line plot in Fig. 1.9 of Tutorial #2. (Minor differences between your results and those figures are acceptable. We do not expect perfect a replication for a turbulent flow.)

_____

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

(i) Follow Step 7 in Tutorial #1 to enlarge the "side pipe". Instead of 0.75 *in* as used in the tutorial, increase the <u>radius</u> of the pipe to $R = 1.5$ *in* (i.e., increase the <u>diameter</u> of the pipe to 3.0 *in*). To ensure a smooth connection between the side pipe and the main pipe, when the radius of the small pipe is increased to 1.5 *in* one should also <u>increase the "depth" of extrusion</u> in step "x." in p. 1-15 of Tutorial #1. The recommendation is to change it from 4.0 *in* to 6.0 *in*. Fig. 1 shows what the modified geometry should look like.
(ii) Set the inlet velocity associated to the "main pipe" to $U_x = 0.05$ m/s; Set the inlet velocity associated to the "side pipe" to $U_y = 0.05$ m/s.
(iii) Instead of "pressure outlet", set the boundary condition for the outlet to "outflow". Under this setting, there will be no need to set the temperature of back flow.
(iv) Instead of liquid water, fill the system with *engine oil*. Use the existing data from Fluent database to set the physical properties of engine oil. The density, specific heat, viscosity, and thermal conductivity of the fluid are all set to constant like in Tutorial #1.
(v) 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 velocity magnitude and temperature in the plane of symmetry, in the fashion of Figs. 1.21 & 1.22 (or 1.25 & 1.26) in Tutorial #1.

Tips:

(i) Following Tutorial #1, the mesh for the modified system should still have enhanced resolution along the wall; see Fig. 2. The mesh shown in Fig. 2 is the minimum requirement. A further refinement of the mesh is strongly recommended. (Otherwise, the relatively low resolution in the interior of the pipe will affect the quality of the line plots for Task 2.)

(ii) There are various options in Results and CFD-Post to adjust the contour plots. An acceptable option (useful for saving ink for the report) in CFD-Post is to replace filled contours with unfilled contours, i.e., just contour lines with no color fill in between. A contour plot in gray scale (black and white) is acceptable only if it is clearly made.
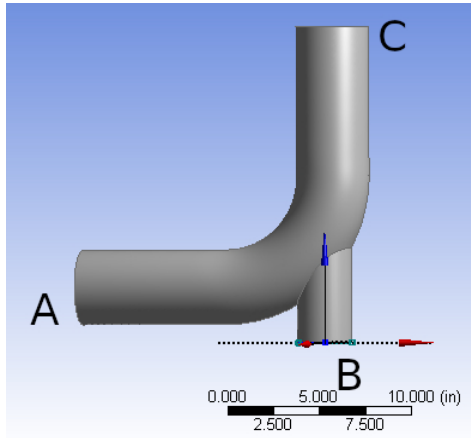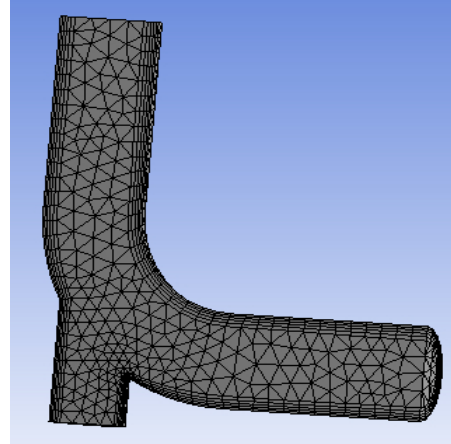
Fig. 1

Fig. 2

Task 2

From the simulation in Task 1, produce the profiles (line plots) of temperature and the velocity component normal to the surface of outlet (not to be confused with "velocity magnitude") along the line, *AB,* as indicated in Fig. 3. 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 *XY 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.)

(ii) The style and color of the line(s) can be edited in *XY plot*. Alternatively, one can "dump" the data to a text file, then use Matlab to do the plotting. The data file can be created by selecting "write to file" in *XY plot*.
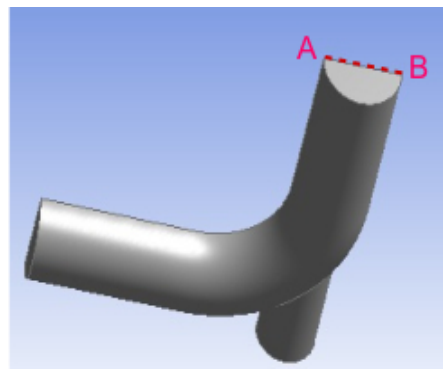

Fig. 3

Task 3

[The detail of this task was explained in the in-class demo in Lecture 5.]

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

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

where ρ and c are density and specific heat of engine oil, v_n is the velocity component normal to the surface of the inlet or outlet, and T is temperature. The integral is performed 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 not uniform across the surface of the outlet. We require that the integral be performed by combining Custom Field Function and Surface Integral utilities in Fluent, as demonstrated in Lecture 5.

As also explained in Lecture 5, while Fluent has a built-in function (under Results:Fluxes) to directly extract the "rate of heat transfer" across a surface, the quantity calculated by that function is different from that defined by Eq. (1). (Further explanations will be given in future lectures.) DO NOT use that built-in function for this task.

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.

Note (for background only): Since there is no internal heat source for this system and density is set to constant, by energy balance we expect the sum of the values of H for the two inlets to be very close to the value of H for the outlet. A very minor discrepancy might still exist due to errors that arise from numerical approximation (discretization, etc.) and possibly the effect of molecular heat conduction at the inlet/outlet. For example, the later effect could be non-zero if the outlet is set to "pressure outlet", which would necessitate the imposition of temperature for back flow. This would allow a nonzero temperature gradient (normal to the surface of the outlet) to remain in the steady state, implying non-zero heat conduction through the outlet. This effect is not included in Eq. (1). Nevertheless, this effect should be suppressed when the boundary condition of the outlet is set to "outflow".