MAE 494/598 Applied CFD, Fall 2019   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 beginning of 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.)
Instructor will provide supplementary tutorials to fill the gaps in the older tutorials.
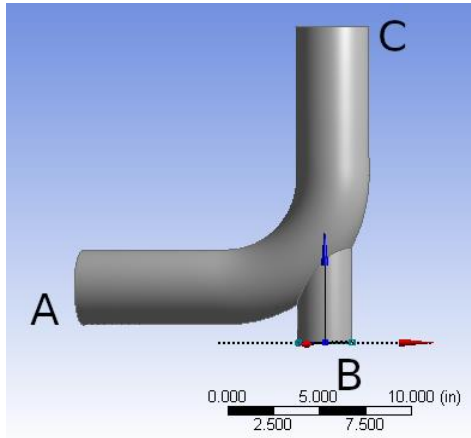

Task 1
Repeat the task in Tutorial #1 but with the following modifications:

(i) Follow Step 7 in Tutorial #1 to enlarge the "side pipe". Instead of 0.75 *in* as used in the tutorial, increase the radius of the pipe to $R = 1.5$ *in* (i.e., increase the diameter 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 increase the "depth" of extrusion 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 $U_x = 0.1$ m/s; Set the inlet velocity associated to the "side pipe" to $U_y = 0.15$ m/s.
(iii) Set the inlet temperature associated to the "main pipe" to $T = 295$ °K; Set the inlet temperature associated to the "side pipe" to $T = 285$ °K; Set the backflow temperature for the outlet to $T = 290$ °K.
(iv) Instead of liquid water, fill the system with *liquid kerosene*. Use the existing data from Fluent database to set the physical properties of liquid kerosene. The density, specific heat, viscosity, and thermal conductivity of the fluid are all set to constant like in Tutorial #1.
(v) Keep the "k-epsilon" model as in Tutorial #1. For simplicity, just use the default setting for the turbulence model and default setting for turbulence-related parameters in the boundary conditions.

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:

Following Tutorial #1, the mesh for the modified system should still have enhanced resolution along the wall; as illustrated in Fig. 2. The actual mesh used in this homework should have a finer resolution that shown in Fig. 2. (Using a low resolution will affect the representation of viscous effect near the wall. Moreover, using low resolution in the interior of the pipe will affect the quality of the line plots for Task 2.)

Fig. 1 The modified geometry.
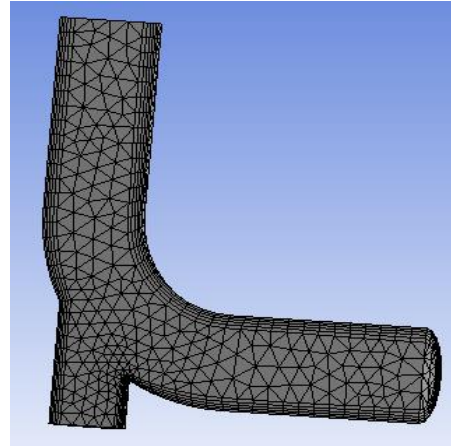
Fig. 2 Enhancement of resolution near the wall. (Note: The actual resolution used in the simulation should be finer that shown here.)
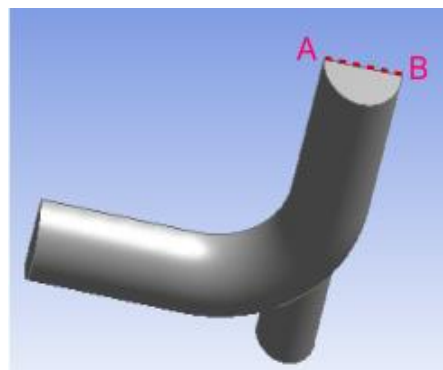
Task 2

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 write the data to a text file, then use Matlab or Excel to do the plotting. The data file can be created by selecting "write to file" in *XY plot*.

Fig. 3 The line *AB* for Task 2.

Task 3

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

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

where ρ and c are density and specific heat of liquid kerosene (as given in Fluent database), 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 (denoted as "S" in Eq. (1)) of the inlet or outlet. Thus, H has the unit of power, in Watts. Be aware that v_n and T are not uniform across the surface of the outlet. A possible way to perform the integral for the outlet is to first define the integrand by Custom Field Function, then use Surface Integral in Report-Fluxes to complete the integration.

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). Please DO NOT use that built-in function for this task.

(Further explanations on the key points for this task will be given in upcoming lectures.)

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 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, which is not covered by the flux of static energy in Eq. (1).