

MAE 460/560 Applied CFD, Fall 2021 Homework 1 (5 points)

Note: 1 point \approx 1% of the total score for the semester.

The report for this assignment should be uploaded to Canvas as a single PDF file. Deadline at Canvas is 11:59 PM (Arizona local time), Friday, September 17th.

No restriction on collaboration for this homework. *If the assignment is completed with the help from other student(s), it should be acknowledged in the beginning of the report. Although collaboration is allowed, the write-up of the report must be done independently.*

Task 0 (for self-exercise only; **no points** and **no needs 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 a perfect replication for the turbulent flow.)

Instructor will provide supplementary in-class tutorials to fill the gaps in the older tutorials.

-
- **Students in both MAE460 and MAE560 sections need to complete Task 1-3.**
 - **For this homework, the simulation may be performed using either the “full pipe” geometry, or “half pipe” geometry by invoking symmetry. Please state your choice in the report.**

Task 1

Repeat the main simulation 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 $R = 1.0$ in (i.e., increase the diameter of the pipe to 2.0 in). To ensure a smooth connection between the side pipe and the main pipe, when the radius of the small pipe is increased to 1.0 in one should also increase the "depth" of extrusion in step "x." in p. 1-15 of Tutorial #1. For example, change it from 4.0 in to 5.0 in. Figure 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 $U_y = 0.15$ m/s.
- (iii) Set the inlet temperature associated to the “main pipe” to $T = 285$ °K; Set the inlet temperature associated to the “side pipe” to $T = 300$ °K; Set the backflow temperature for the outlet to $T = 290$ °K.
- (iv) Instead of liquid water, choose *liquid kerosene* as the material for the fluid. Use the existing data from Fluent database to set the physical properties of the liquid. The density, specific heat, viscosity, and thermal conductivity of the fluid are all set to constant.
- (v) Choose turbulence k - ϵ model. Otherwise, use the default setting for that model. Also, use the default setting for boundary conditions associated with turbulence model. (Note that the default for the 2021 R2 version of Ansys-Fluent is now the k - Ω model. For this exercise, it needs to be changed. In some older versions of the software, k - ϵ model is already the default.)
- (vi) For the creation of mesh, simply follow the in-class demo in Lecture 3 to set *Element Size* to 0.5 cm, and set *Inflation* to *Program Control*. Bypass other steps in the original tutorial. (As noted in Lecture 3, we expect to have an enhanced mesh resolution along the wall, but not along the plane of symmetry.)
- (vii) For this homework (which is only concerned with steady solution), it suffices to use *Hybrid Initialization*.

With the modifications detailed in the preceding page, run the simulation to seek *steady solution*. The key deliverables for Task 1 are:

(D1) 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.

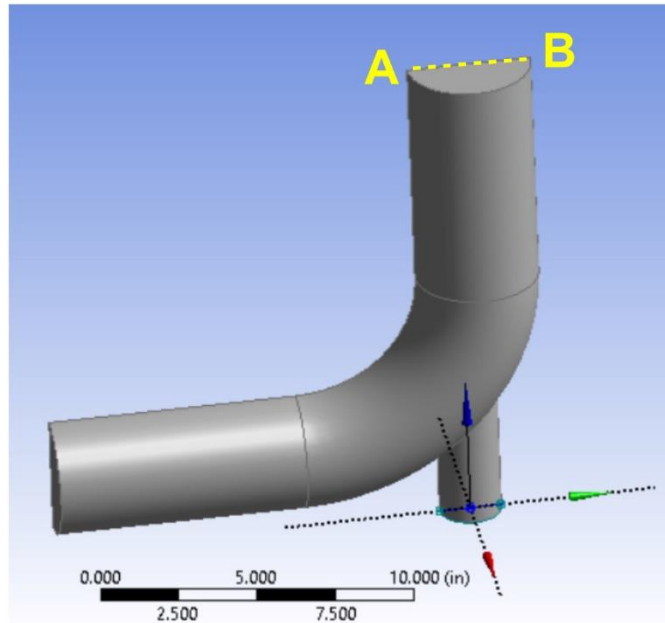


Fig. 1 The modified geometry. The line \overline{AB} is defined for Task 2.

Task 2

From the simulation in Task 1, produce the following additional deliverables:

(D2) The line plots of the profiles of *temperature* and the *velocity component normal to the surface of outlet* (not to be confused with "velocity magnitude") along the line, \overline{AB} , as indicated in Fig. 1. 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 temperature and velocity. Please clearly label the abscissa and ordinate of the plot.

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 absolute coordinates of points *A* and *B*. (The "new surface" is defined as the line that passes through the two points.)

(ii) The style and color of the line(s) can be edited in *XY plot*. Alternatively, one can output the data to an external file, then use Matlab or Excel to make the plot. The data file can be created by selecting "write to file" in *XY plot*.

Task 3

From the simulation in Task 1, 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

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

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 (see Tutorial #2), 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 will be given in upcoming lectures.)

The key deliverables for Task 3 are:

(D3) The three values of H (in Watts) associated to the two inlets and one outlet, and a brief description of the procedure that is used to compute those values.

Note: (i) If the simulation is performed using the “full pipe” geometry, the value of H will be twice that of the “half pipe” case. Please indicate which geometry is chosen for the calculation. (ii) 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 convective flux of static energy in Eq. (1).