

MAE 460/560 Applied CFD, Fall 2022 Homework 1 (5 points)

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

Upload the report to Canvas as a single PDF file.

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.*

Task 0 (for self-exercise only; **no points** and **no needs to show results**)

Go through 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 1-4.**
 - **For this homework, the simulations 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 (see (vi)-(vii) for minor deviations from the tutorial) 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. For example, change it from 4.0 in to 6.0 in. Figure 1 shows what the modified geometry should look like.
- (ii) Set the inlet velocity associated to the "main pipe" to $U_x = 0.01$ m/s; Set the inlet velocity associated to the "side pipe" to $U_y = 0.03$ m/s.
- (iii) Set the inlet temperature associated to the “main pipe” to $T = 300$ °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, choose *liquid diesel* 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 *Laminar* model.

As discussed in class, the following settings (which deviate from the original Tutorial #1) are acceptable:

- (vi) For the creation of mesh, simply set *Element Size* (and *Max Size* under *Sizing*) to 0.5 cm, and set *Inflation* to *Program Control*. Bypass other steps in the original tutorial. (See in-class demos provided by instructor.) Note that this is by no mean the optimal setting, but a compromise given the limit on the number of cells we can use.
- (vii) For this homework, it suffices to use *Hybrid Initialization*.

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

- (a) 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 and our in-class demos, 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 than shown in Fig. 2. (Using a lower resolution will affect the representation of viscous effect near the wall. Moreover, using a lower resolution in the interior of the pipe will affect the quality of the line plots for Task 2.)

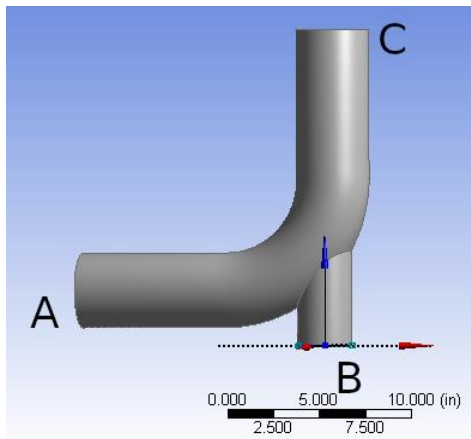


Fig. 1 The modified geometry.

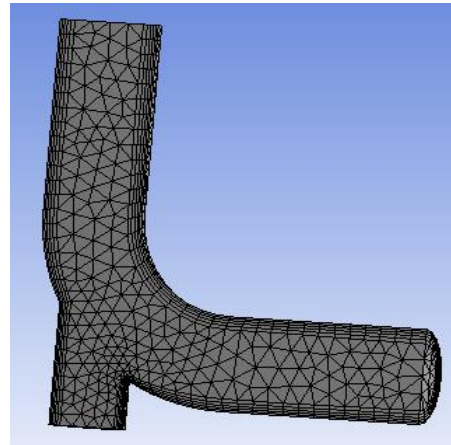


Fig. 2 Enhancement of resolution near the wall. (The actual resolution used in your work should be finer than shown here.)

Task 2

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

(b) 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, *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 temperature and velocity. Please clearly label the abscissa and ordinate of the plot.

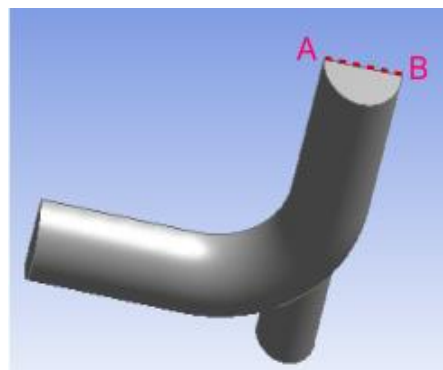


Fig. 3 The line *AB* for Task 2.

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 diesel* (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 the lectures.)

The key deliverables for Task 3 are:

(c) 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).

Task 4

Repeat Task 1 but modify a few parameters:

- (i) Set the inlet velocity associated to the "main pipe" to $U_x = 0.2$ m/s; Set the inlet velocity associated to the "side pipe" to $U_y = 0.4$ m/s.
- (ii) Choose *k-epsilon* turbulence model instead of *Laminar* model. For simplicity, just use the default setting for the turbulence model, and default setting for turbulence-related parameters in the boundary conditions.

The deliverables for this task are:

(d) 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.

[Disclaimer: The designated setting for this task is not an endorsement of the *k-epsilon* model. Turbulence model is a complicated subject which this class will only briefly cover. We designate the use of *k-epsilon* model and its default setting just so there is a uniform standard for grading.]