

Problem Description

A cold fluid flows into a right-angled pipe through a large inlet area and mixes with a warmer fluid that flows through an additional smaller inlet area at the bend of the pipe. We utilize ANSYS Fluent to analyze the overall velocity and temperature outcomes for two different setups in a series of three tasks.

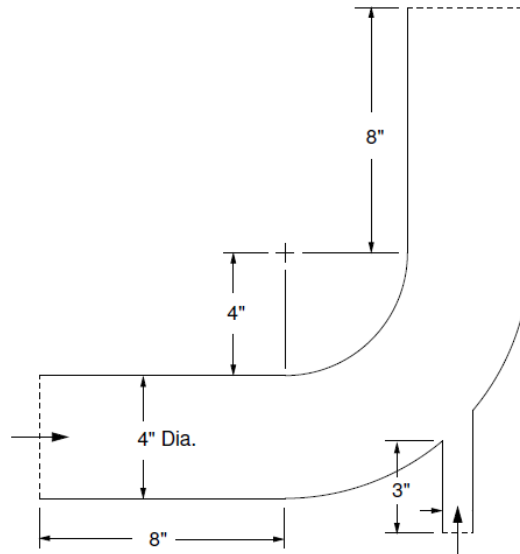


Figure 1. General Problem Schematic¹

Task 0

Assigned Tutorial #1 and #2 were completed as a self-exercise. Note the following initial parameters:

Large inlet diameter:	4"
Large inlet fluid velocity:	0.4 m/s
Large inlet temperature:	20 °C
Large inlet turbulent intensity:	5%
Small inlet diameter:	1"
Small inlet fluid velocity:	1.2 m/s
Small inlet temperature:	40°C
Small inlet turbulent intensity:	5%

No results to show.

¹ Modified from ANSYS Fluent Tutorial 1 Figure 1.1

Task 1

Another analysis was performed with modified initial parameters per the following:

- Small inlet diameter: 2"
- Small inlet fluid velocity: 0.6 m/s

The steady state temperature and velocity results were analyzed under the new parameters and contour plots were generated, providing a visual representation of the results.

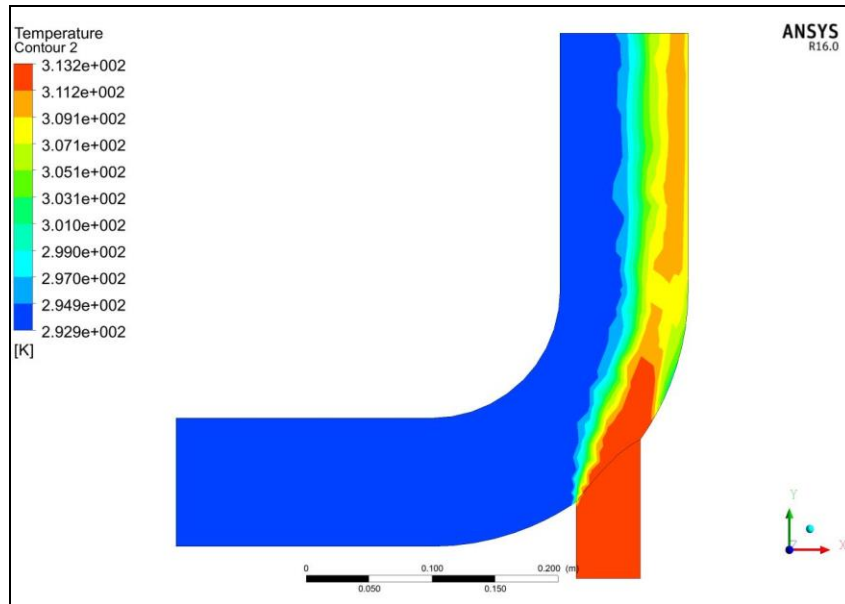


Figure 2. Temperature Contour Plot about Symmetry Plane

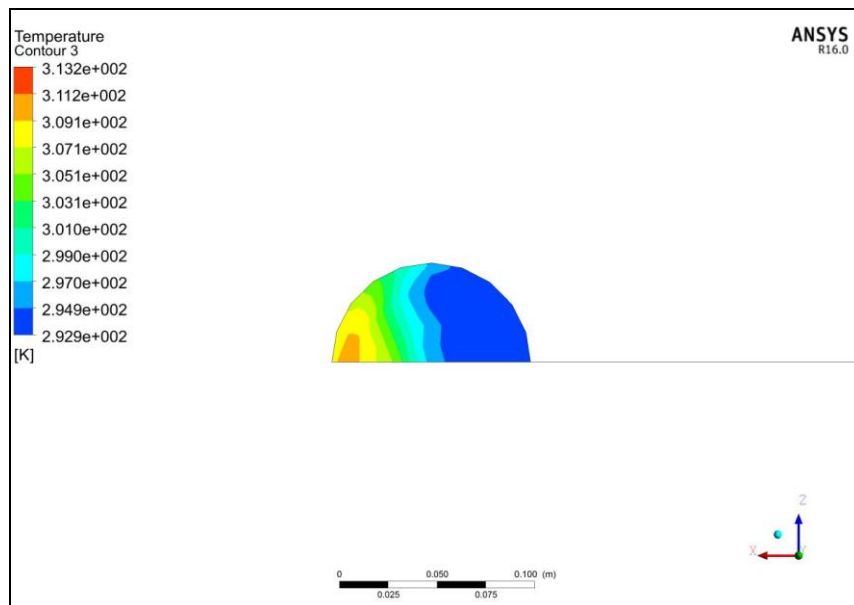


Figure 3. Temperature Contour Plot about Outlet Plane

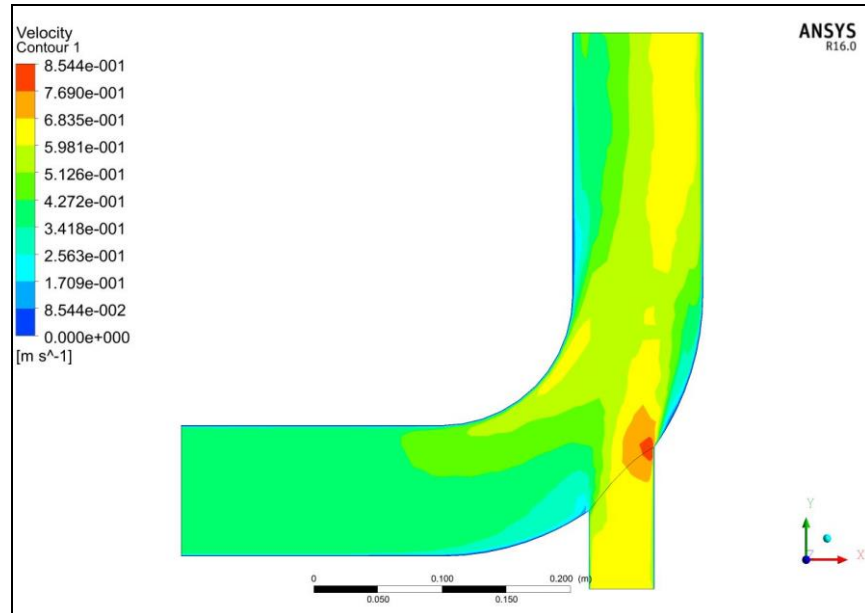


Figure 4. Velocity Contour Plot about Symmetry Plane

Task 2

Line plots of temperature and velocity along the centerline of the outlet (line “AB”), parallel to the symmetry plane, were generated. The line plots show the two different results from the varying parameters of Task 0 (red) and Task 1 (blue).

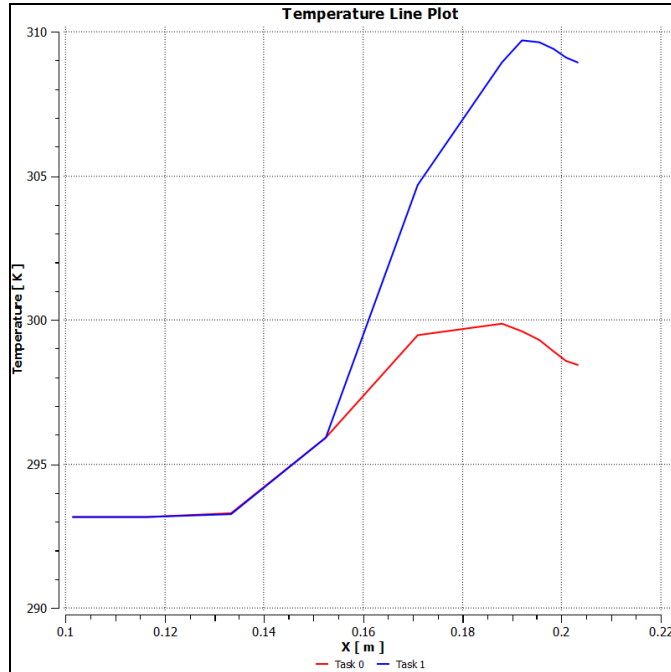


Figure 5. Temperature Line Plot about Line “AB”

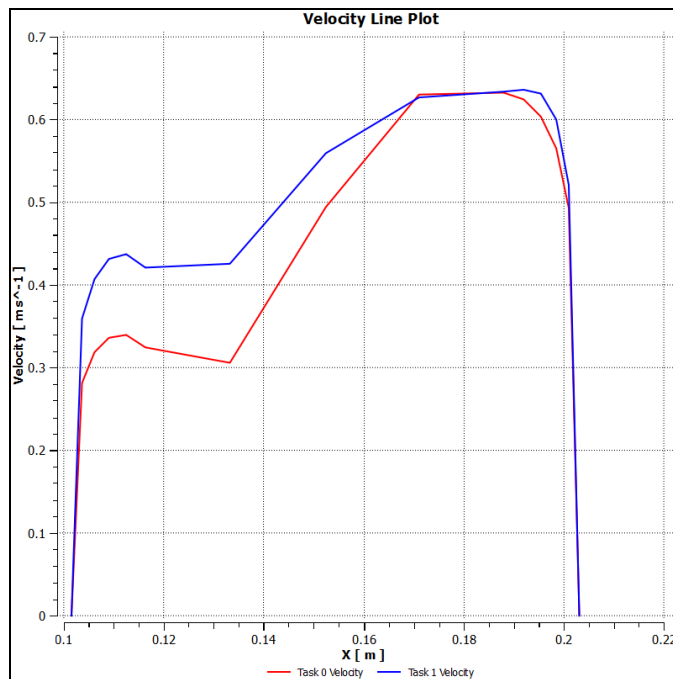


Figure 6. Velocity Line Plot about Line “AB”

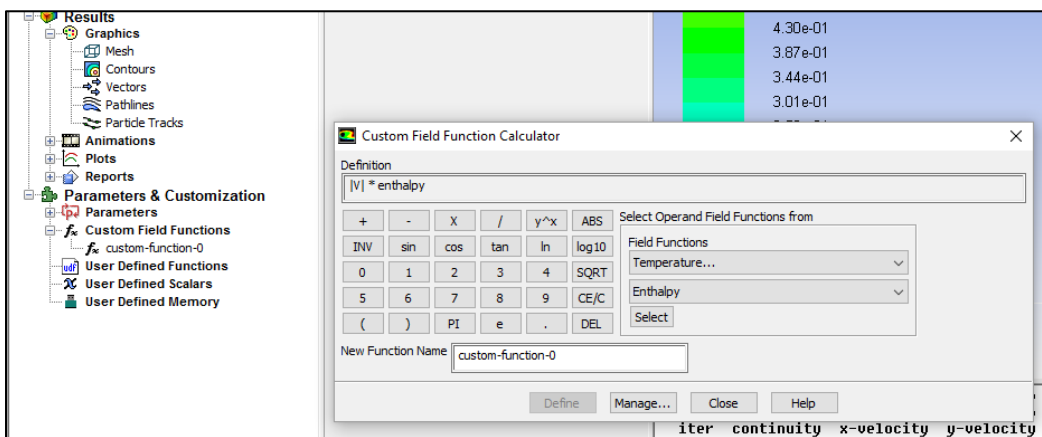
Task 3

Checked the energy balance and mass balance of our system considering Task 1 parameters. Because there is no heat source or sink in this system, the energy into the system is equivalent to the energy out of the system. The same applies to the mass flow rate where there is no internal mass source or sink so the mass flow rate into the system is equivalent to the mass flow rate out of the system. Note that throughout the system we assume density (ρ) and specific heat (C_p) of the fluid are constants. Also, velocity (v_n) and temperature (T) are constant only at the inlets.

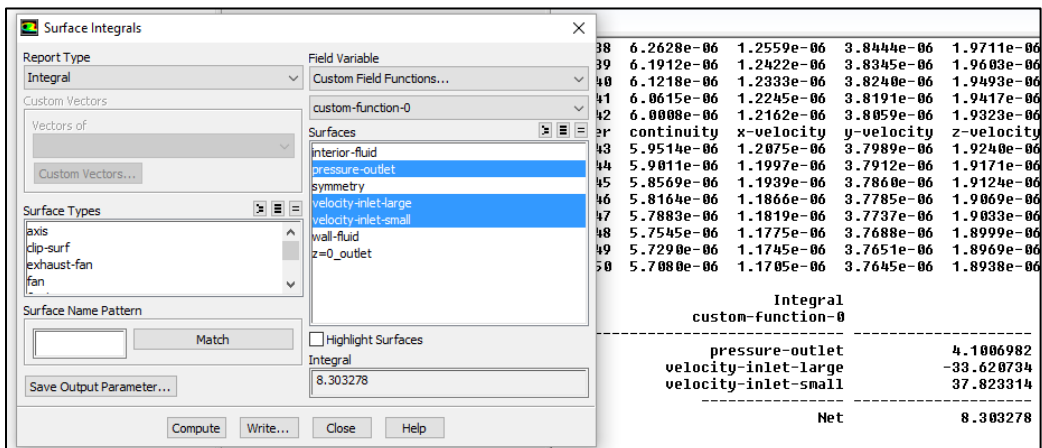
Beginning with the given Eq. (1) for heat through a specific surface, the constants are pulled out of the area integral (this only applies at the inlets).

$$H \equiv \iint_A v_n \cdot \rho \cdot C_p \cdot T \, dA \quad \text{Eq. (1)}$$

Then, using ANSYS Fluent's "Custom Field Function" option, a function of velocity multiplied by temperature is created.



Using the surface integral, the custom function created earlier is integrated over the specified area.



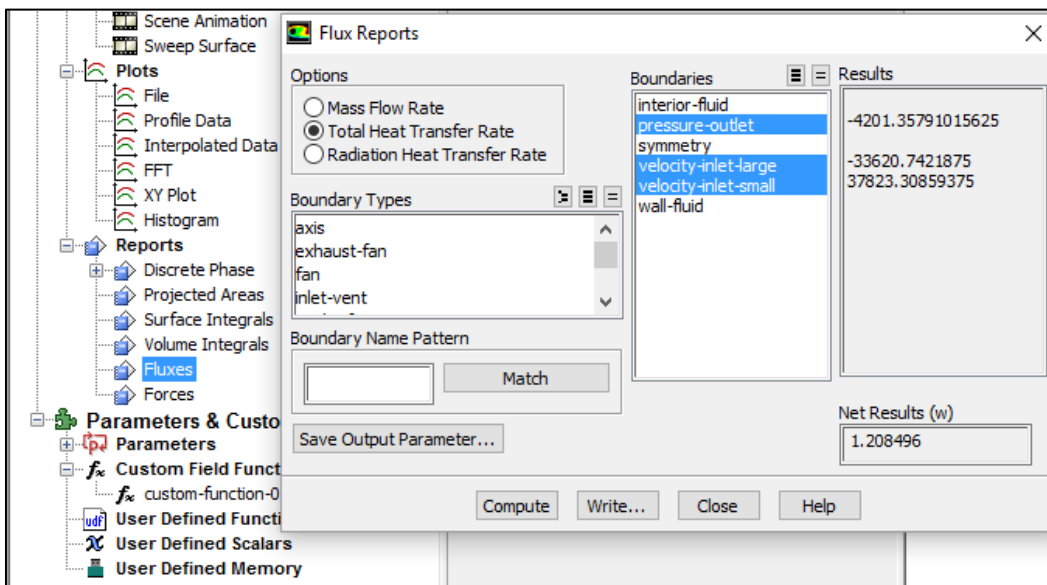
The resulting value is then multiplied by the density to obtain the flow rate of heat through each surface.

$$H_{inbg} = 1000 \frac{kg}{m^3} \cdot 33.621 \frac{J \cdot m^3}{kg \cdot s} = \mathbf{33621 W}$$

$$H_{inms} = 1000 \frac{kg}{m^3} \cdot 4.101 \frac{J \cdot m^3}{kg \cdot s} = \mathbf{4101 W}$$

$$H_{out} = 1000 \frac{kg}{m^3} \cdot 37.823 \frac{J \cdot m^3}{kg \cdot s} = \mathbf{37823 W}$$

The final values for the heat can be compared to other built-in ANSYS Fluent evaluation methods² under Results>Reports>Fluxes>Total Heat Transfer>Compute.



Note, it is important to recognize that when using ANSYS Fluent's flux report, ANSYS assigns positive and negative signs based on flow (heat or mass) going into or out of the system. In the heat flow case, heat is going into the system at the small inlet where temperature is the highest and is leaving out of the system at the large inlet and large outlet where temperature is the lowest.

Using the given Eq. 2 for mass flow rate, because the density and velocity are constant at the pipe inlets, they can be pulled out of the area integral (note this only applies at the inlets).

$$M \equiv \iint_A v_n \cdot \rho \, dA \quad \text{Eq. (2)}$$

$$M = v_n \cdot \rho \cdot A$$

² Credit to MAE 598 Applied Computational Fluid Dynamics Challenge Question 1 solution by Vikram Suresh

Then, performing a hand calculation can be performed. As mentioned earlier, in this case, the mass flow rate at the outlet is equal to the sum of the mass flow rates at the inlets.

$$M_{inbg} = 0.4 \frac{m}{s} \cdot 1000 \frac{kg}{m^3} \cdot \frac{0.0508^2 \pi}{2} m^2 = 1.621 \frac{kg}{s}$$

$$M_{inasm} = 0.6 \frac{m}{s} \cdot 1000 \frac{kg}{m^3} \cdot \frac{0.0254^2 \pi}{2} m^2 = 0.608 \frac{kg}{s}$$

$$M_{out} = M_{inbg} + M_{inasm} = 1.621 + 0.608 \frac{kg}{s} = 2.229 \frac{kg}{s}$$

The hand calculated values can then be compared to the values calculated by ANSYS Fluent's flux reports again.

