

## TASK – 1A

For this case of modeling a cylindrical shaped water heater, we use Solid works software considering the simplicity. We then import the geometry to Ansys workbench and perform further analysis. All the units are assumed consistent to the MKS unit system. The orientation of the figure is such that it aligns with figure attached to the problem statement, with the cylinder axis running along Y axis and origin at the center of the bottom surface. We call three named surfaces the inlet, the outlet and bottom. The inlet is set as velocity-inlet with a constant velocity of 0.06 m/s and temperature of 288.15 K (15°C). The outlet is set as outflow and the bottom as wall with temperature at constant 318.15 K (45°C).

### Deliverable 1:

A plot of the mesh along the plane of symmetry before and after refinement in a pancake shaped region, height 0.15 m from origin along Y axis is shown below. Also, for this meshing we adopt a minimum element size of 0.0155 m and have enabled adaptive sizing and inflation to program controlled.

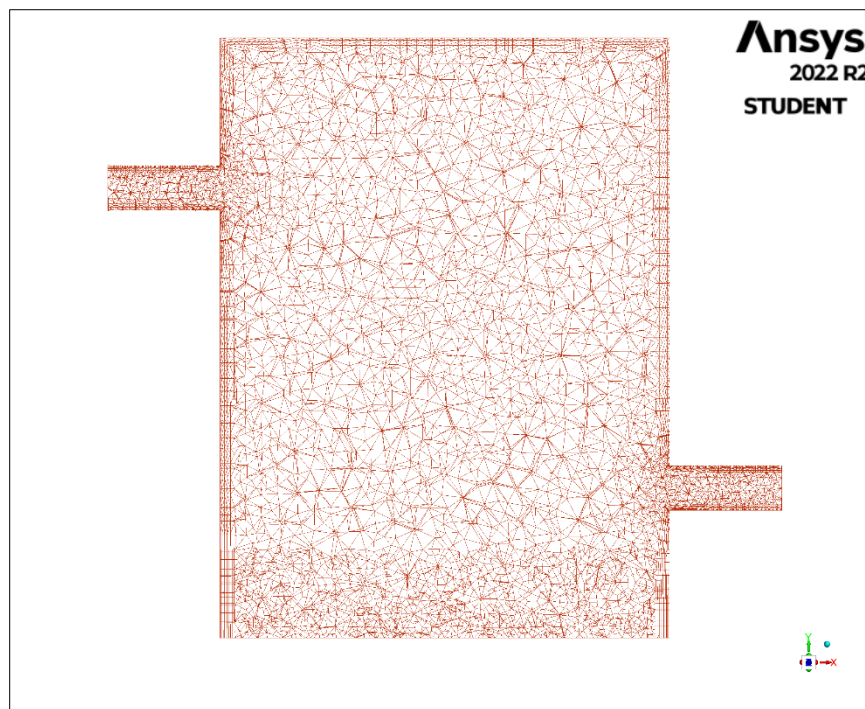


Figure 1: Plot showing the mesh after refinement in the cylindrical bottom region.

The operating temperature is calculated by taking the average of both the inlet and outlet temperature i.e., 288.15 K (15°C) and 318.15 K (45°C) which turns out to be 303.15 K (30°C). For calculating density at this temperature, we use the equation detailed in the NIST journal excerpt attached to the statement. Also, by since Coefficient of thermal expansion depends on the density of the fluid, we find the relation between as follows:

$$\Delta V = V \cdot \Delta T \cdot \beta$$

Where  $\Delta V$  is the change in volume,  $V$  the initial volume,  $\Delta T$  the change in temperature and  $\beta$  the coefficient of thermal expansion. For differential changes of volume and temperature we can rearrange the above relation and use the following equation for volume change per unit temperature change.

$$\frac{dV}{dT} = V \cdot \beta$$

Also,  $V = (M, \text{mass}) / (\rho, \text{density})$ . Remember mass is constant.

$$V = \frac{M}{\rho} \Rightarrow \frac{dV}{dT} = \frac{-M}{\rho^2} \cdot \frac{d\rho}{dT}$$

Substituting this value in equation for volume change per unit temperature,

$$\frac{dV}{dT} = \frac{M}{\rho} \cdot \beta = \frac{-M}{\rho^2} \cdot \frac{d\rho}{dT}$$

Solving for  $\beta$  from the above equation we get,

$$\beta = -\frac{d\rho}{dT} \cdot \rho$$

We use the above equation to generate a subroutine in PyCharm which directly computes the required values. The value of density at operating temperature of 303.15 K (30°C) is calculated to be 995.644871364 kg/m<sup>3</sup> and the coefficient of thermal expansion is 0.0003032827120239392 K<sup>-1</sup>.

### Subroutine to compute the density and Coefficient of thermal expansion from NIST data:

```
from sympy import *
# Import all the necessary libraries

t = Symbol('t')
rho = 999.85308 + 6.32693*(10**-2)*t - 8.523829*(10**-3)*(t**2) +
6.943248*(10**-5)*(t**3) - 3.821216*(10**-7)*(t**4)
print("The NIST equation for density in terms of temperature t is \n", rho)
# Defining and printing the density values as per NIST

rho_diff = rho.diff(t)
# Differentiating the density function with respect to temperature for later
calculations

exp_coef = -(1/rho)*rho_diff
print("The equation for thermal expansion coefficient is \n", exp_coef)
# Defining and printing the thermal expansion coefficient function

f1 = lambdify(t, rho)
# Defining the NIST density function for calculations

print("The value of density of water at 30 deg celsius is: ", f1(30))
# Finding the density value at 30 deg celsius, the average temperature (NIST
uses celsius not Kelvin units)

f2 = lambdify(t, exp_coef)
# Defining the expansion coefficient function for calculations

print("The value of thermal expansion coefficient at 30 deg celsius is: ",
f2(30))
# Finding the expansion coefficient at 30 deg celsius, the average
temperature
```

### Output:

```
1 C:\Users\rakkarak\PycharmProjects\AML\HW1\venv\
Scripts\python.exe C:\Users\rakkarak\AppData\Roaming\
JetBrains\PyCharmCE2022.2\scratches\scratch_1.py
2 The NIST equation for density in terms of temperature
t is
3 -3.821216e-7*t**4 + 6.943248e-5*t**3 - 0.008523829*t
**2 + 0.0632693*t + 999.85308
4 The equation for thermal expansion coefficient is
5 -(-1.5284864e-6*t**3 + 0.00020829744*t**2 - 0.
017047658*t + 0.0632693)/(-3.821216e-7*t**4 + 6.
943248e-5*t**3 - 0.008523829*t**2 + 0.0632693*t + 999
.85308)
6 The value of density of water at 30 deg celsius is:
995.644871364
7 The value of thermal expansion coefficient at 30 deg
celsius is: 0.0003032827120239392
8
9 Process finished with exit code 0
```

## Deliverable 2:

The Ansys setup is configured for liquid water with Boussinesq approximation and choosing density and thermal expansion coefficient as calculated before. The meshing of the model was performed multiple time with varied element sizes and changing adaptive sizing settings. At very small mesh element sizes we find that the outlet temperature plot is not smooth and converges at certain iteration number. If the iteration number increases the graph further fluctuates. This could be due to the truncation error resulting due to refined mesh sizes. After executing the analysis multiple times for a pressure based, steady state solution we arrive at a more reasonable solution. A contour plot of the y-velocity in the plane of symmetry for such a solution is attached below.

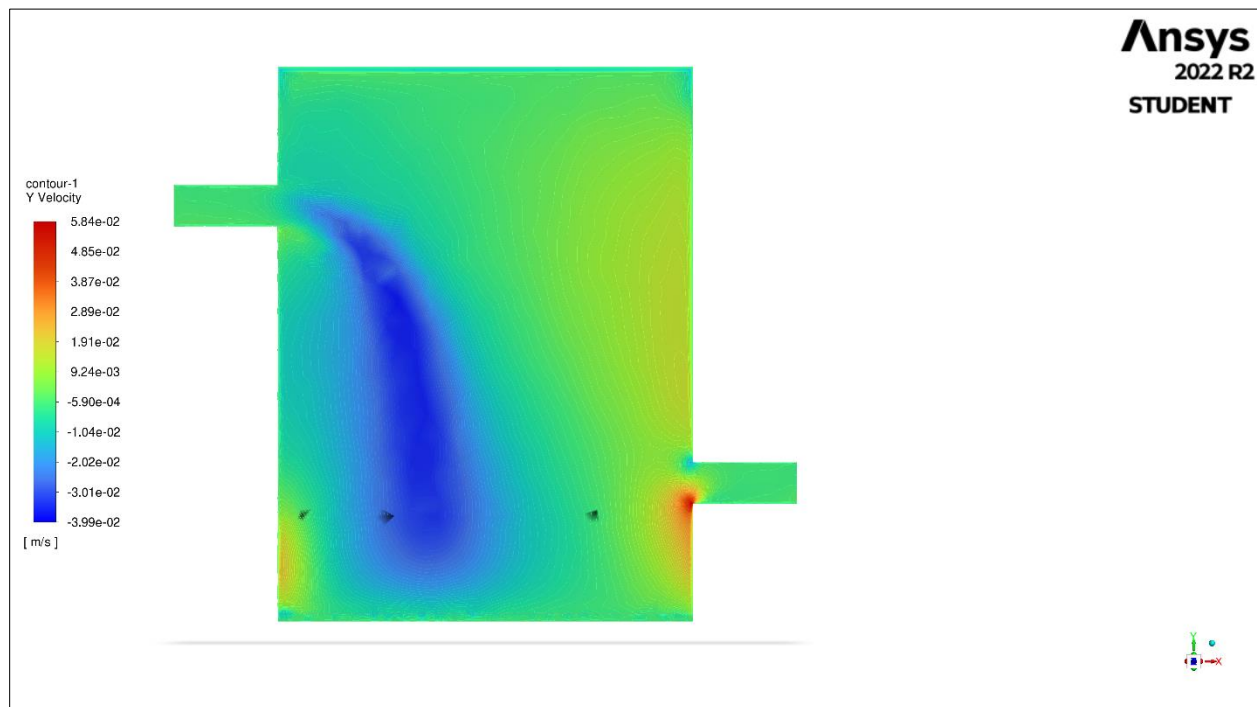


Figure 2: Contour plot showing the Y component velocity along symmetric half plane in XY plane.

This contour plot is reasonable, since both inlet and outlet where X component velocity dominates Y component is very negligible. Note that corresponding to the flow direction in the negative Y direction we have a bluish path with negative Y component velocities which occupy the lower parts of the velocity variation spectrum.

### Deliverable 3:

A contour plot of temperature in the plane of symmetry for the steady solution is plotted similarly, shown below. We have modified the temperature ranges and color palette for the contrast to be visible. Also, the plot is zoomed in for the stark temperature difference at its bottom to be evident. Most of the fluid enters at lower temperature 288.15 K (15°C) as can be notes at the inlet and the outlet temperature is roughly 293 K indicating a rise in temperature of outflowing water. As can be expected the boundary temperature at the bottom surface is 318.15 K (45°C) as was prescribed by us during analysis.

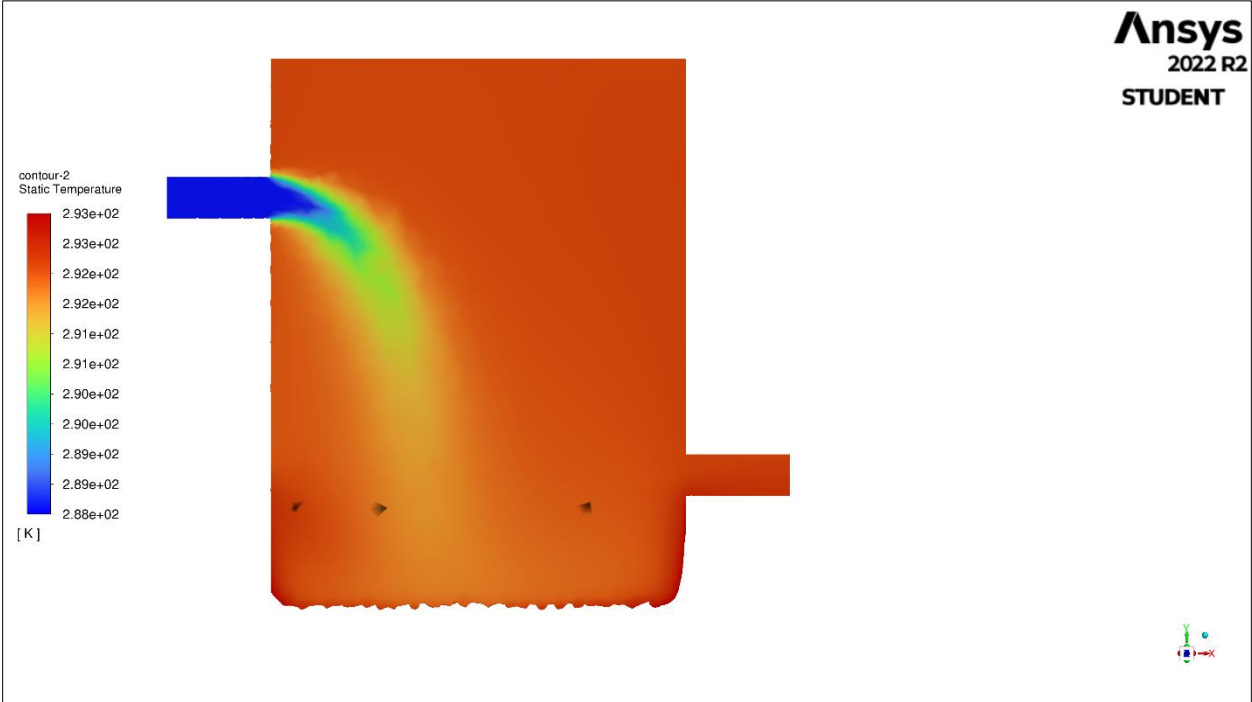


Figure 3: Contour plot showing variation of temperature along the symmetry plane along XY.

## Deliverable 4:

The average weighted temperature at outlet over the cross-sectional surface is calculated by defining a report definition value of outlet temperature and is taken as the condition to check convergence as was mentioned in the statement. The following details of the iteration number and temperature was noted from the console:

Iter	avg-outlet	time/iter
1100	2.9131e+02	0:03:11 100
1200	2.9134e+02	0:00:00 0
1300	2.9137e+02	0:00:00 0

As is evident, the convergence criterion of 0.1 temperature difference over 100 iterations was noticed first by 1200 iterations and just to confirm the convergence since the graph fluctuates close to the convergence, we run it for 100 more iterations. The convergence criterion is still fulfilled hence we can confirm that the Average weighted temperature value at outlet to be 291.3 K.

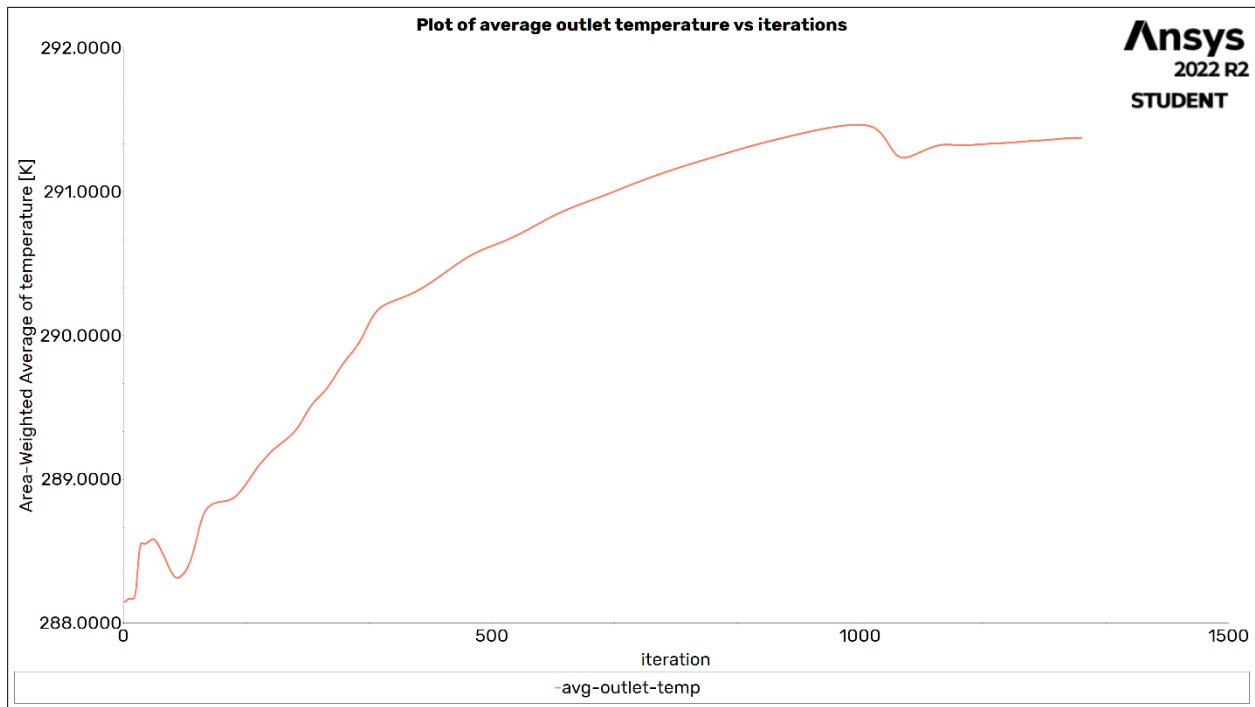


Figure 4: Line plot of the outlet temperature as a function of the number of iterations.

## TASK - 1B

For this task we repeat the task 1a with the gravity turned off thus pronouncing the effect of buoyancy alone. We follow the same steps as the task 1a and calculate the deliverables using the same convergence criteria.

### Deliverable 5:

Like task 1a we witness convergence at 1400 iterations first, but we run it till 1500 to confirm the convergence, same can be witnessed from the temperature outlet plot.

iter	avg-outlet	time/iter
1300	2.8969e+02	0:05:00 200
1400	2.8974e+02	0:02:44 100
1500	2.8975e+02	0:00:00 0

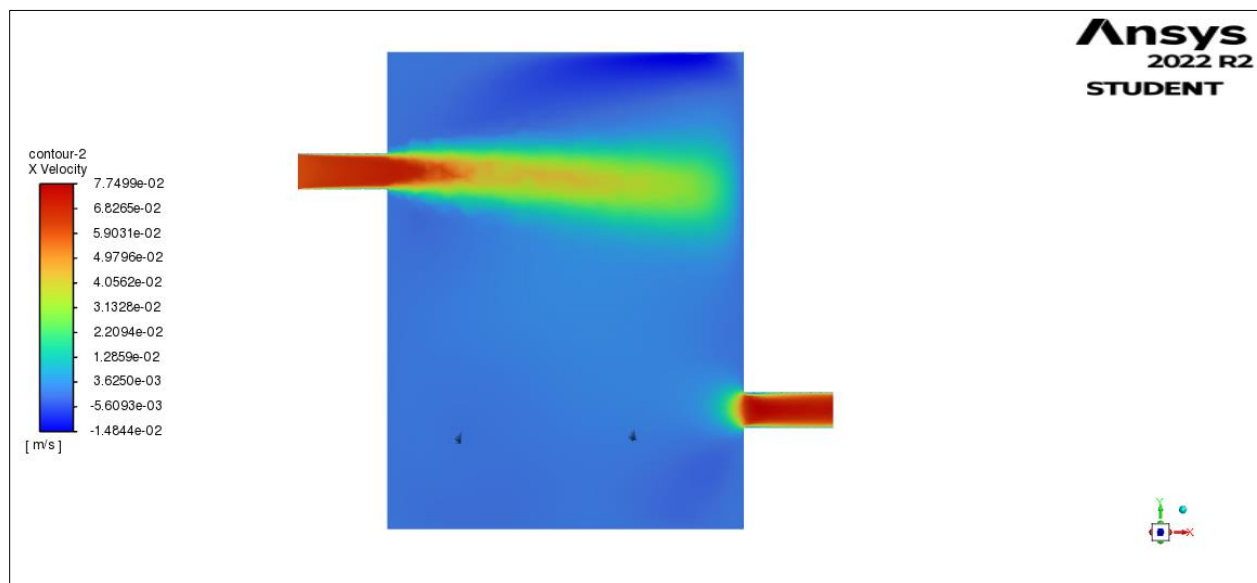


Figure 5: Contour plot of X velocity component in the plane of symmetry XY for steady solution

The contour plot shows that since the gravity is turned off the flow has a longer range of projectile which is reasonable since the downward force will be lesser comparatively.

## Deliverable 6:

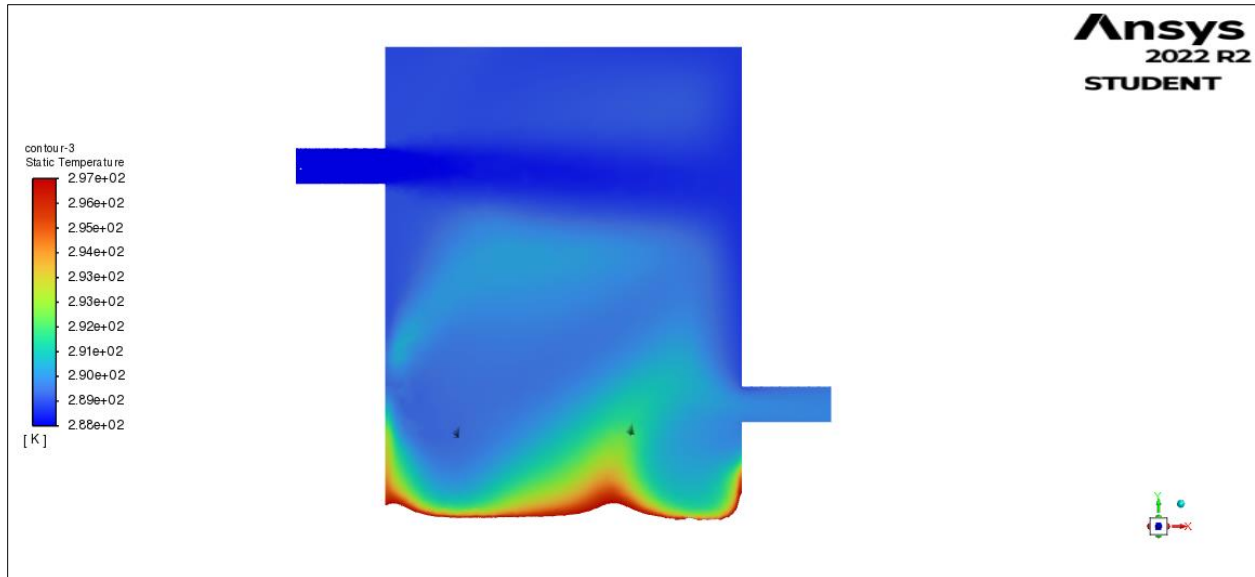


Figure 6: Contour plot of temperature in the plane of symmetry XY for steady solution

The contour plot of temperature shows that the heat dissipation is higher compared to the previous case. This could possibly be due to the momentum of the flow getting disrupted and the fluid having higher time for heat conduction at the bottom plate. Further the velocity contour could be referred to confirm out speculations.

## Deliverable 7:

The value of area weighted average of temperature over the outlet surface is found out to be 289.7 K.



## TASK – 1C

In this task we retain all the settings of task 1a i.e., we check the gravity on and continue our calculation of a transient solution rather than steady state one. We notice the existence of time step, iteration per time step and other report definitions in the report definitions tab auto populated. Also, we conduct this simulation in two steps first till 15 mins interval to extract the contour plot then for the remaining 45 mins, to finish the 1-hour time span.

### Deliverable 8:

For this simulation owing to the huge time span and high computation time we choose a time step size and iterations per time step modestly. This choice could be a better approximation to the case when multiple runs were conducted in task 3 for estimating at which time step size where truncation errors might creep in. We adopt a time step size of 15 secs and run it for 240 number of steps at 12 iterations per step maximum.

### Deliverable 9:

The plot of outlet temperature i.e., the weighted average over the outlet as a function of time over the whole range is extracted at the end of 1 hour (end of second step) and is attached below:

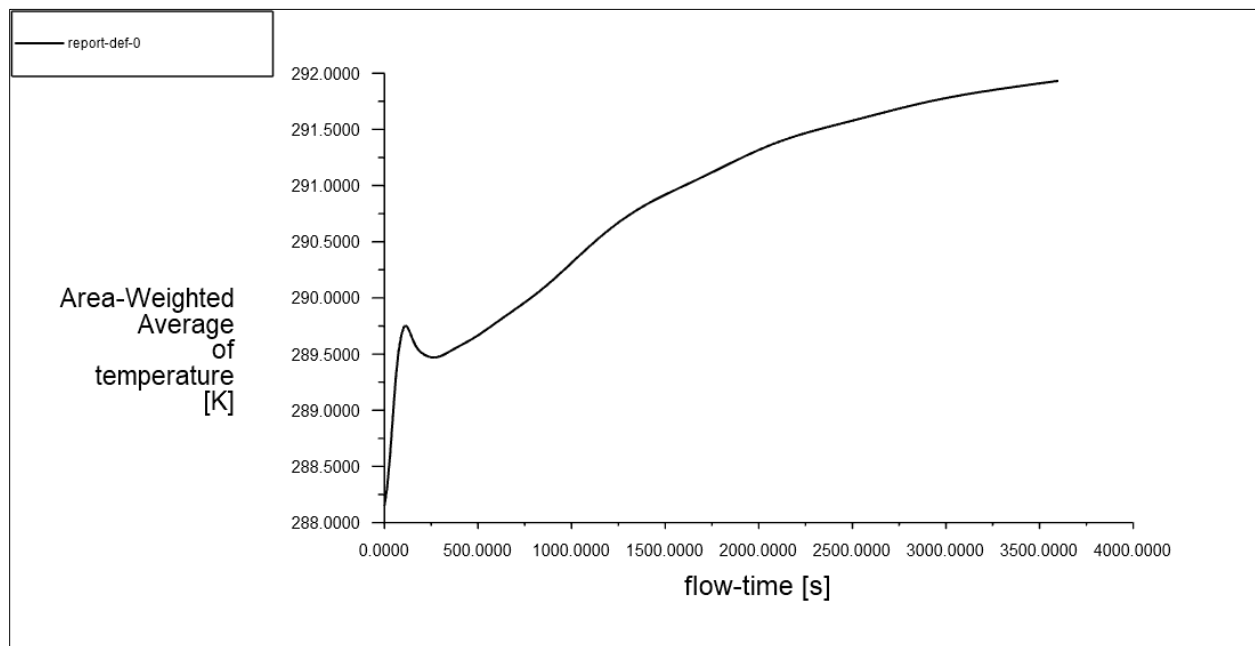


Figure 7: Plot of weighted average temperatures at outlet vs flow time.

**Deliverable 10:**

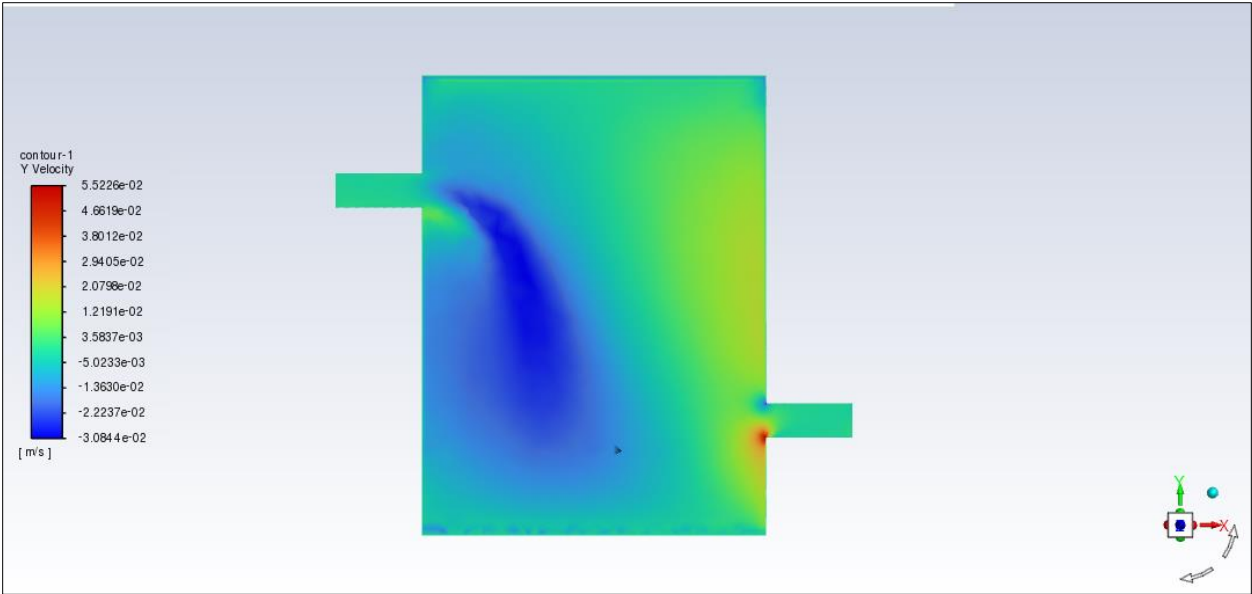


Figure 8: Contour plot of Y velocity in the plane of symmetry XY at the end of 15 minutes.

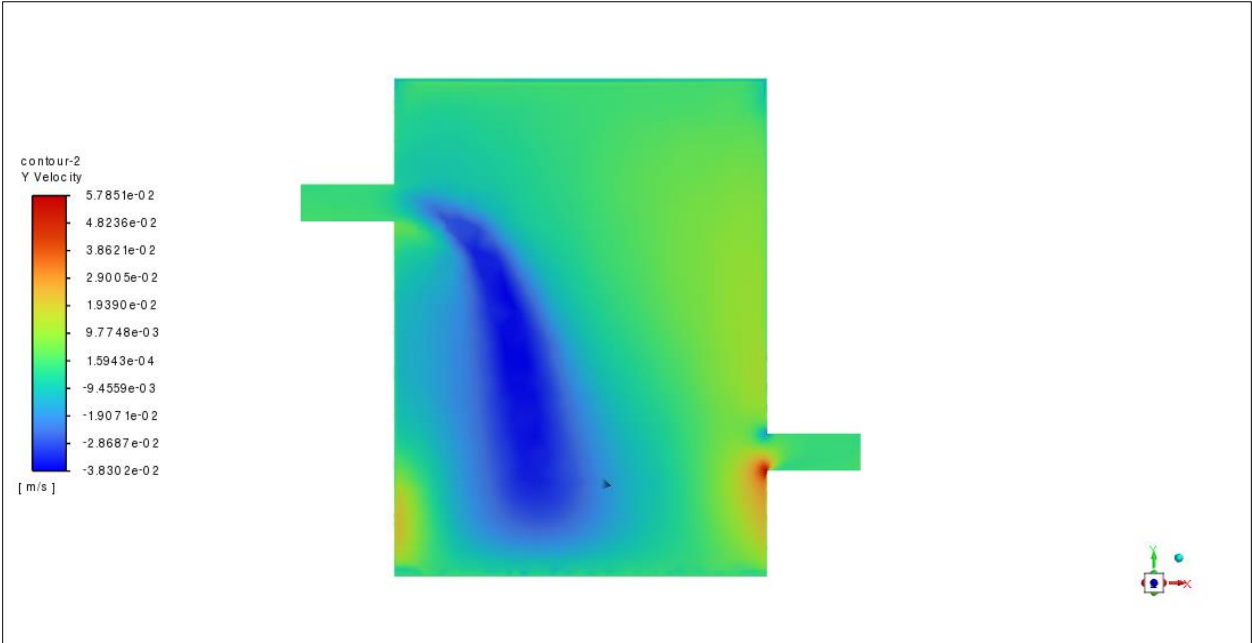


Figure 9: Contour plot of Y velocity in the plane of symmetry XY at the end of 60 minutes.

At the end of 15 minutes, we see that the flow is closer to the wall of the vessel but as time goes on the velocity dies down and the flow with higher range is noticed.

## TASK - 2A

Like task 1 we model the spiral pipe heater in Solid works and export the geometry to Ansys workbench. We use equation defined curve options to draw the coil, since equations in terms of parameter  $\tau$  are listed in the problem statement and then extrude it with diameter of 4 cm (0.04 m). For this simulation we use constant density option turning gravity off with water liquid as the fluid. Set the boundary conditions for inlet as velocity inlet (with varying velocities each run) and temperature at stable 300 K. For outlet set it as outflow and for the wall set the heat flux value as 700 W/m<sup>2</sup>. We then use laminar model and seek steady state solution for this problem.

### Deliverable 11:

The area weighted average temperature is set as one of the report definitions as in task 1 for outlet and considering constant temperature of 300 K at inlet, we find  $\Delta T = T_{out} - T_{in}$  for each run with varied velocity at the inlet. As in task 1 we use the area weighted average value of temperature as a criterion for convergence, by letting it run for enough iterations. The values for temperature differences at different velocity values calculated and also these values were plotted against velocity value as can be seen from the attachment below:

Inlet velocity (m/s)	$T_{out}$ (K)	$T_{in}$ (K)	$\Delta T = T_{out} - T_{in}$
0.008	312.33	300	12.33
0.016	306.14	300	6.14
0.032	303.14	300	3.14
0.064	301.58	300	1.58

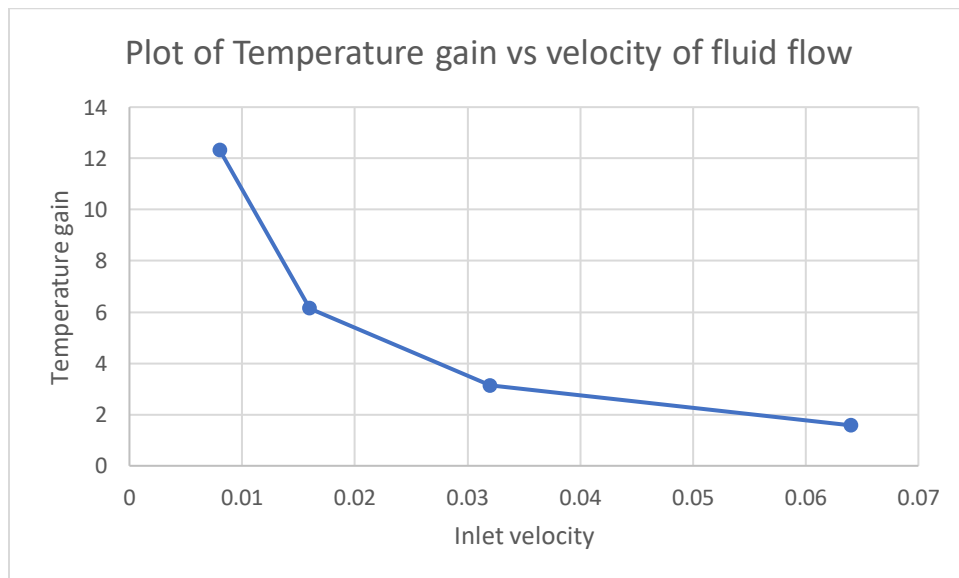


Figure 10: Plot showing the temperature gain as a function of inlet velocity.

The table and plot show that as the velocity of inlet increases, the temperature gain reduces. This makes sense as lower inlet velocities mean slower motion through the pipe and the time available for heat transfer from wall to the fluid is higher while higher velocities mean the opposite hence the temperature gain is lesser in the latter case.

## Deliverable 12:

For the run with inlet velocity 0.016 m/s, contour plots of temperature and velocity magnitude over the circular opening is plotted. The contour intervals, color palette and resolution of the image are adjusted to get a clear picture of the variations.

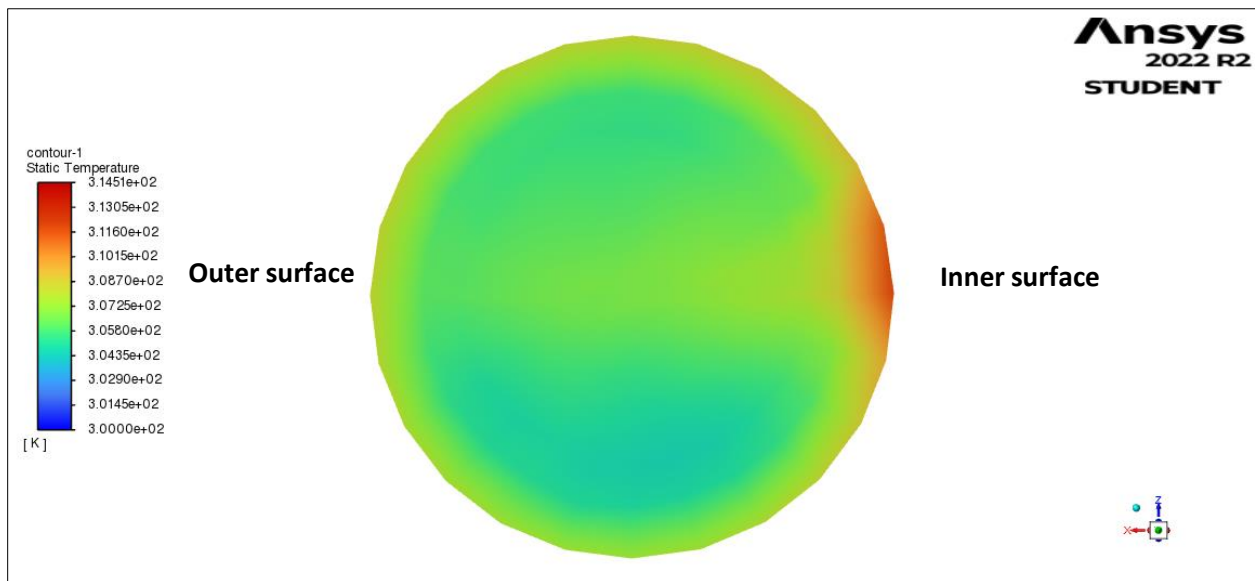
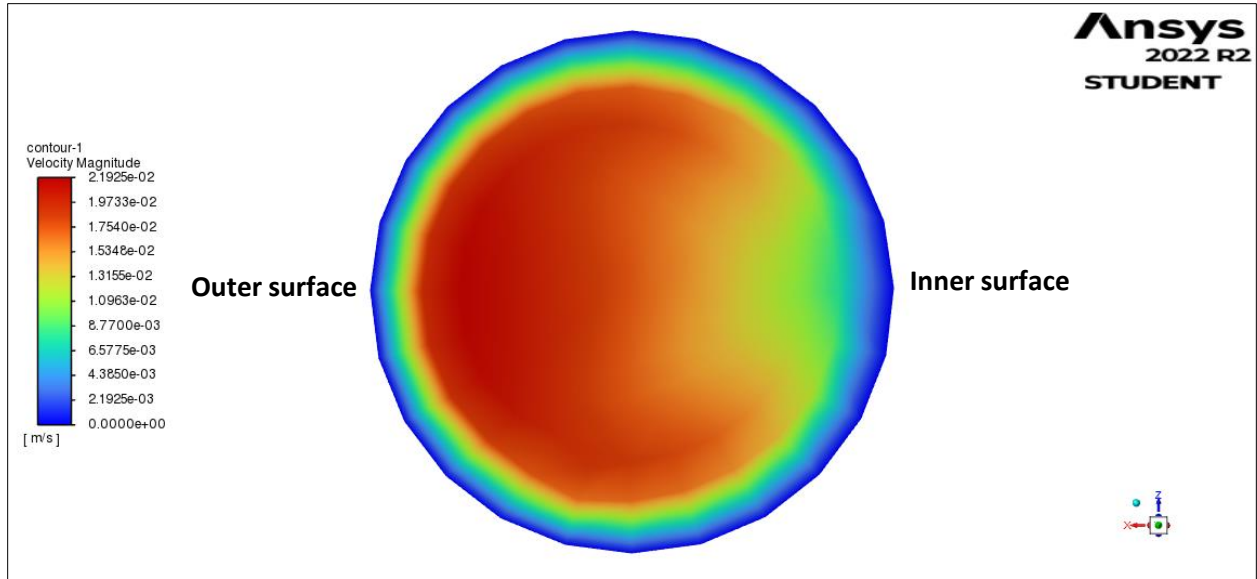


Figure 11: Contour plot of static temperature over the outlet.

Since the fluid circles round the spiral pipe, it is understandable that the fluid's free surface may distort closer to the outer pipe surface due to centrifugal force. We can simply conserve angular momentum of a far-off particle as opposed to a particle at a lesser radius:

$$v = r \cdot \omega$$

Since moment of inertia of a particle under consideration is always constant and so is its angular velocity, particle undergoing angular motion at higher radius should have higher velocity. So, heat will not be transferred to the outer surface fluid particles effectively as compared to inner particles since they have less velocity and high time for heat conduction. The same can be confirmed with the velocity contour plot given below.



## TASK - 2B

For this task we turn off the heating on the walls to depict the viscous heating and so we will enable viscous heating in the laminar model options. Also, we add engine oil from fluent data base and set it as the fluid running through the pipe. We then seek steady state solution for higher set of velocities 1, 2 and 4 m/s, all these to have a more pronounced viscous heating.

### Deliverable 13:

The area weighted average temperature at outlet has been defined as in the previous tasks and the temperature difference for each different inlet velocities has been calculated. The plot of these values with the velocity has been plot and has been attached below.

Inlet velocity (m/s)	$T_{out}$ (K)	$T_{in}$ (K)	$\Delta T = T_{out} - T_{in}$
1	300.04	300	0.04
2	300.1	300	0.1
4	300.24	300	0.24

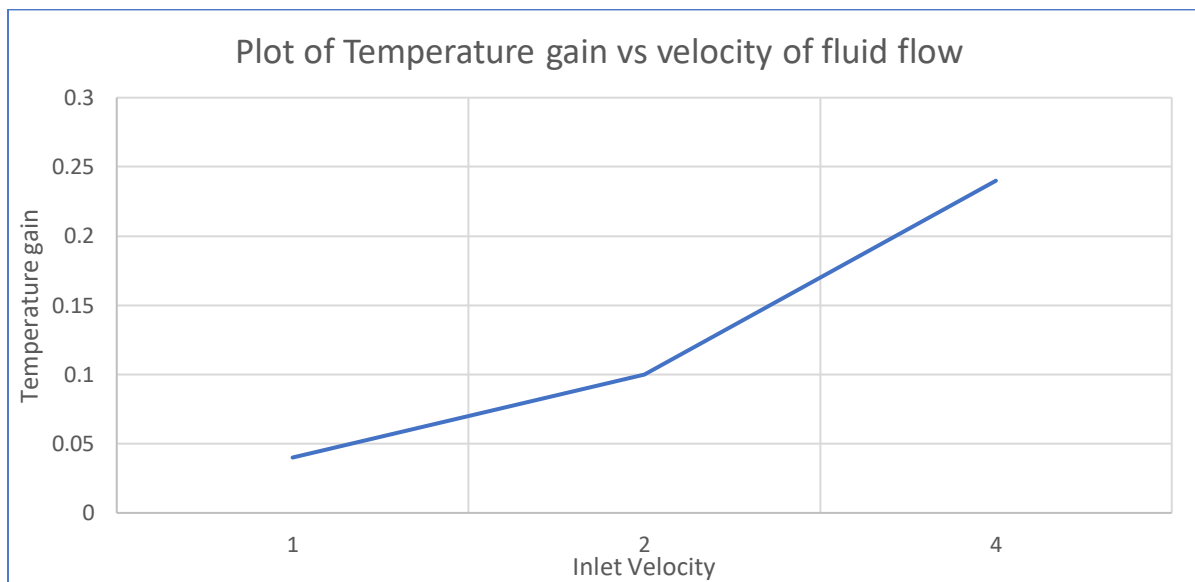


Figure 13: Plot showing the variation of temperature gain due to viscous heating as a function of inlet velocity.

As the inlet velocity increases, the temperature gain also increases which is understandable since the higher velocity in this case increases the viscous heating. One way to understand is that higher velocity means higher kinetic energy which eventually converts to viscous heat between multiple layers of the fluid.

**Deliverable 14:**

For the case with velocity inlet 2 m/s we plot the contour graph of temperature over all exterior surfaces i.e., wall, inlet, outlet. The contour graphs are attached below.

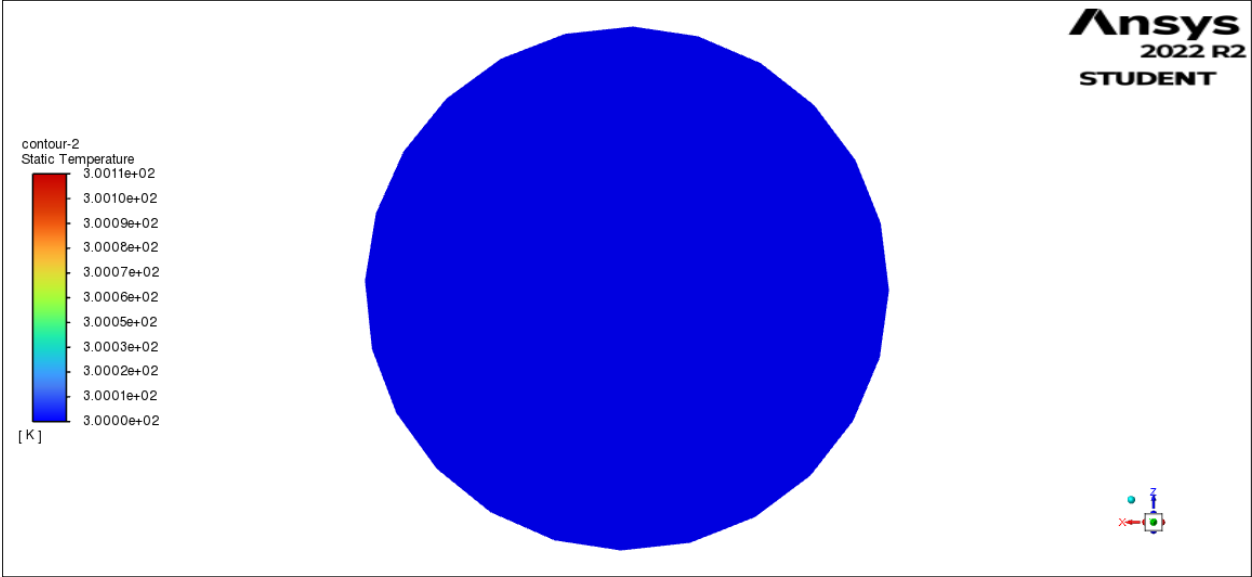


Figure 14: Contour plot of inlet for static temperature.

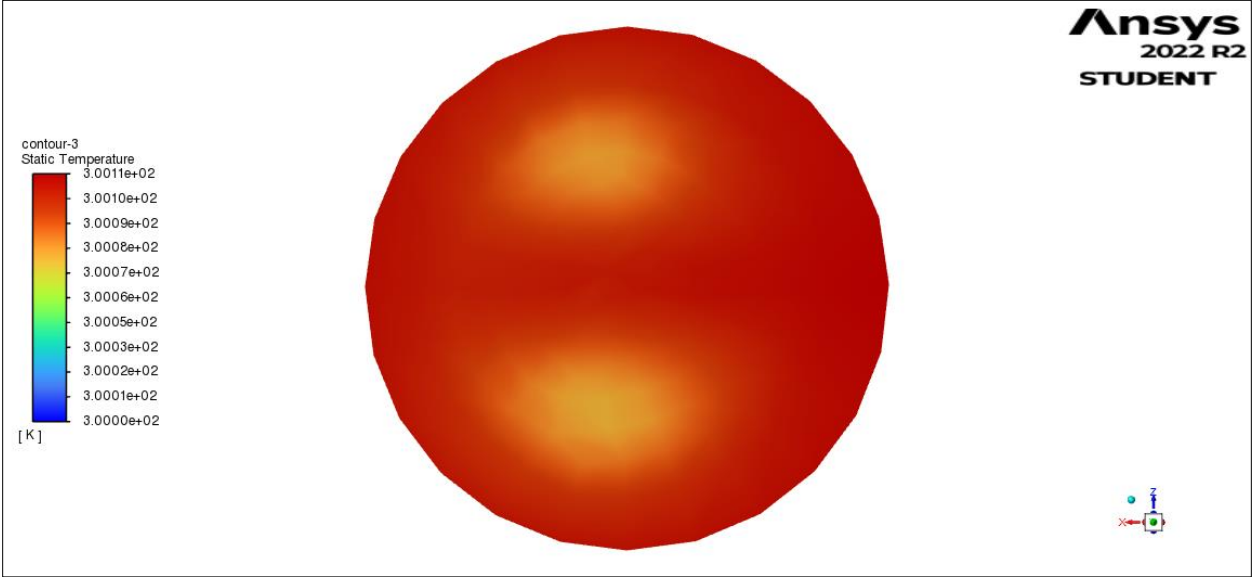


Figure 15: Contour plot of static temperature at outlet.

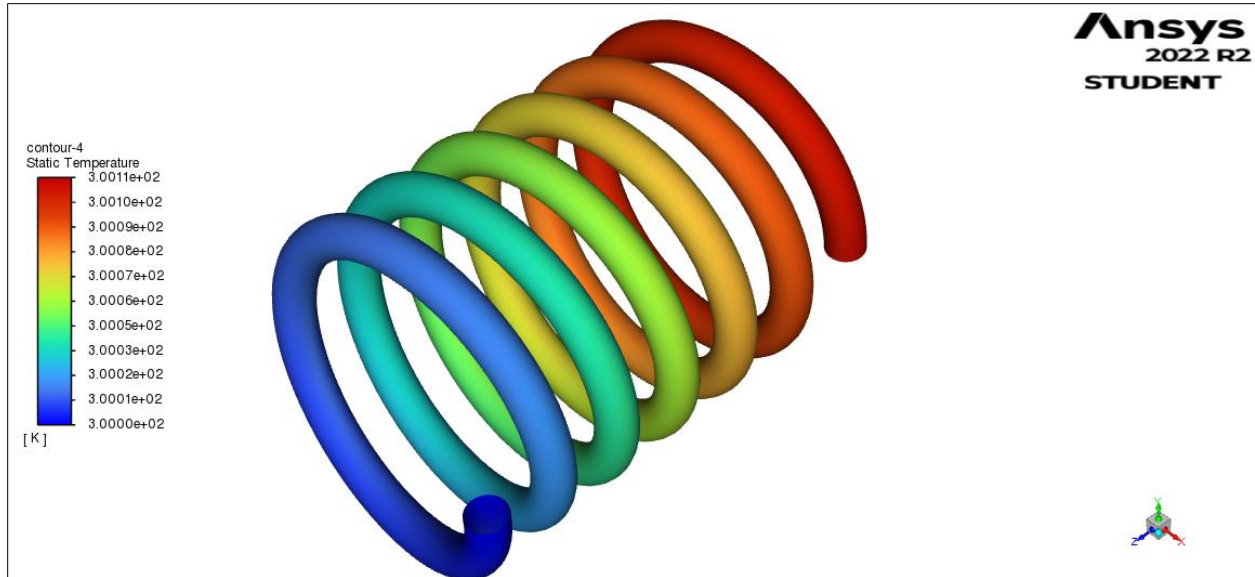


Figure 16: Contour plot of static temperature along the wall surface (panoramic view)

As is evident the viscous heating is prominent in the latter end of the pipe since longer the pipe, longer is the time available for conversion of kinetic energy to heat. Since inlet fluid is supplied at fixed temperature the whole cross section is of singular temperature value. For the outlet cross section, since the viscous heating is expected to be higher at the outer surface (with higher velocity) it has consistent red region as opposed to inner surface. Which is because the inlet surface particles would be of lesser velocity and hence lesser heat generation.



### TASK - 3

For this task we assume the fluid to be air and density to ideal gas with density-based solver while seeking transient solution. We set energy model on with K omega model for default settings. We set the boundary condition for outlet as pressure outlet with total pressure of 10,000 Pa and backflow temperature of 300 K. We also set gauge pressure to zero so that total pressure is identical to operating pressure. To have a pressure difference in chamber A, we define a cell register for the whole volume of chamber A and use patch function with pressure parameter for 10,200 Pa. After making any changes to the settings we make sure to patch after initializing.

#### Deliverable 15:

To arrive at an ideal number of time steps we run the simulation for different numbers starting at 50 and increment it by 20 steps every time. We see a variation in the temperature, density and z-velocity in trends at some critical value of step number as is recorded below:

Run count	Time step size	Number of time steps	Max iterations per time step	Temp (max)	Temp (min)	Rho Max	Rho min	V <sub>z</sub> max	V <sub>z</sub> min
1	0.001	50	20	300.00 49	297.86 04	0.11 8249	0.11 5189	61.8 448	- 7.25 274
2	0.00071 429	70	20	300.01 2	297.92 2	0.11 8049	0.11 5356	58.2 6932	- 9.26 613
3	0.00055 556	90	20	300.06 36	298.08 03	0.11 7874	0.11 5688	52.2 4317	- 8.01 626
4	0.00045 4545	110	20	300.06 74	298.18 98	0.11 7687	0.11 598	45.9 8753	- 5.41 319
5	0.00038 4615	130	20	300.00 9	298.17 5	0.11 7494	0.11 61	41.5 7027	- 4.24 754

The trend reverses at step number of 130 hence we take a value lesser than that. We assume step number to be 124 and the step size to be 0.000403226 seconds. Considering the complexity and time for analysis while increasing iterations per time step we choose a default value of 20 iterations per time step as ideal.

## Deliverable 16:

For this task, we set mesh element sizing to 2 cms and the max element size limiting to 3 cms. We set inflation to be program controlled with no adaptive size. In addition to inflation to have mesh refined locally at the neck area we define mesh size separately for the neck region and reduce it to 1 cm locally. The figure below shows denser mesh elements at the neck region along the plane of symmetry.

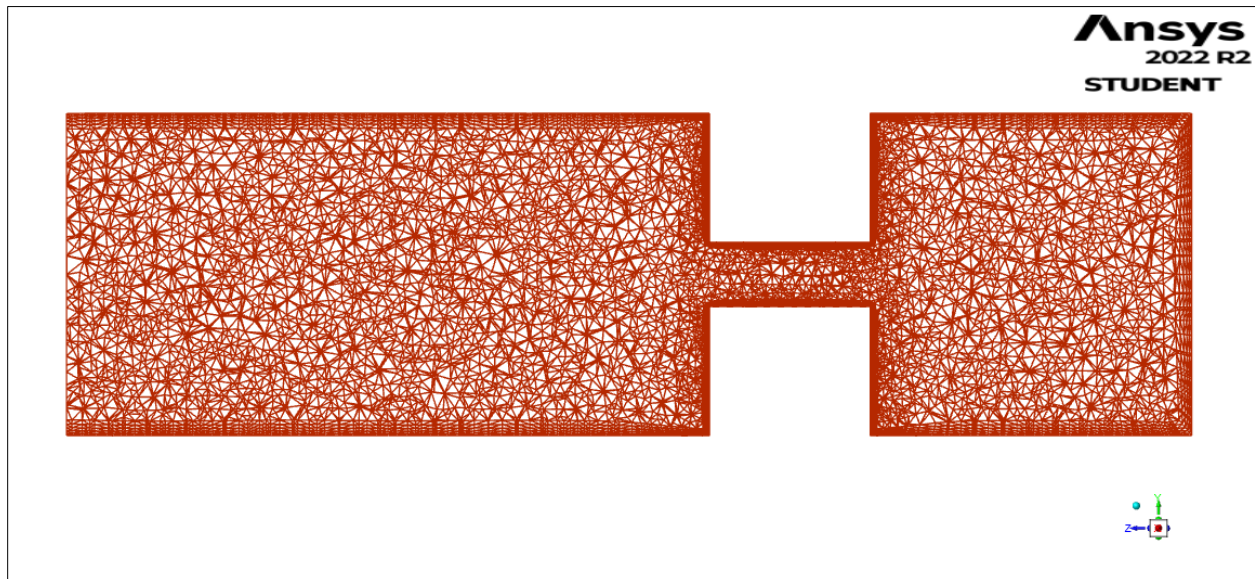


Figure 17: Mesh plot for task 3 with refined mesh around the neck region.

## Deliverable 17:

We plot the contour graphs for temperature, density, and z-velocity at two different time instances  $t = 0.025$  secs and  $t = 0.05$  secs. For this case we run the calculation first till 0.025 secs i.e., time step size of 0.000403226 and for 62 number of time steps. After plotting for all three parameters, we rerun the simulation with 62 more so that the total time comes down to be 0.05. Then we again plot the parameters at this instant. The following are the contour plots resulting.

As can be understood the static temperature of chamber A decreases and this is reasonable because the pressure drops as the air escapes to chamber B (Ideal gas equation:  $P \propto T$ ). Also, it can be noted that the adiabatic expansion of gas in chamber B follows with a decrease in temperature around that region. The density contour graph shows that the chamber A has higher density reasonable because it has higher pressure compared to chamber B. The escaping gases will have a higher velocity along Z axis and the velocity contour makes sense as well.

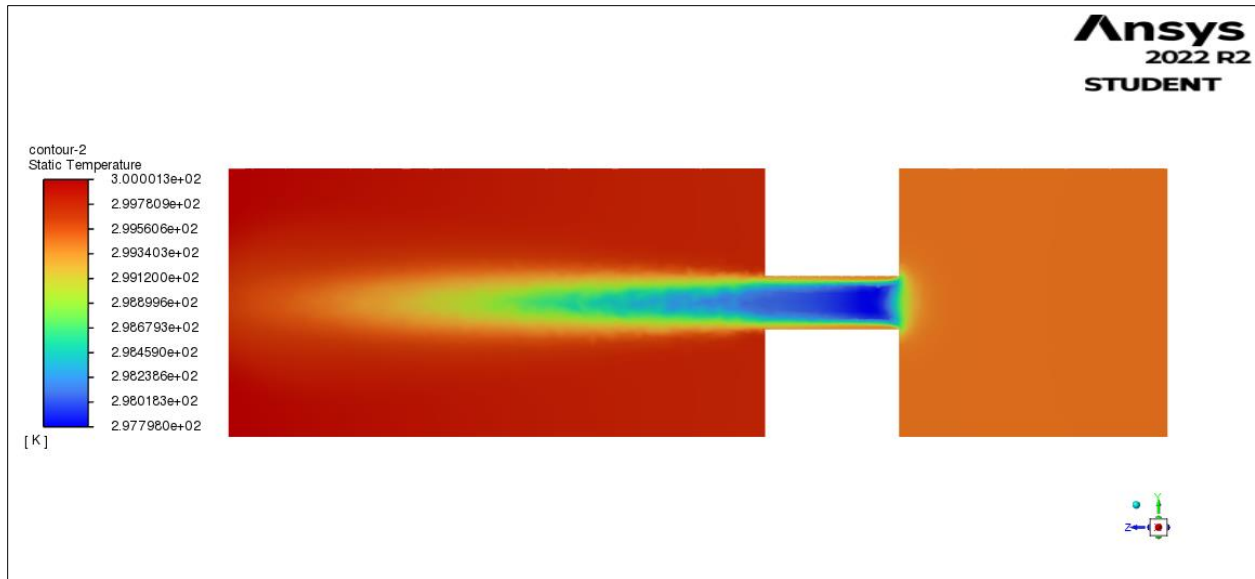


Figure 18: Contour plot of static temperature along symmetric plane YZ at 0.025 seconds of transient simulation.

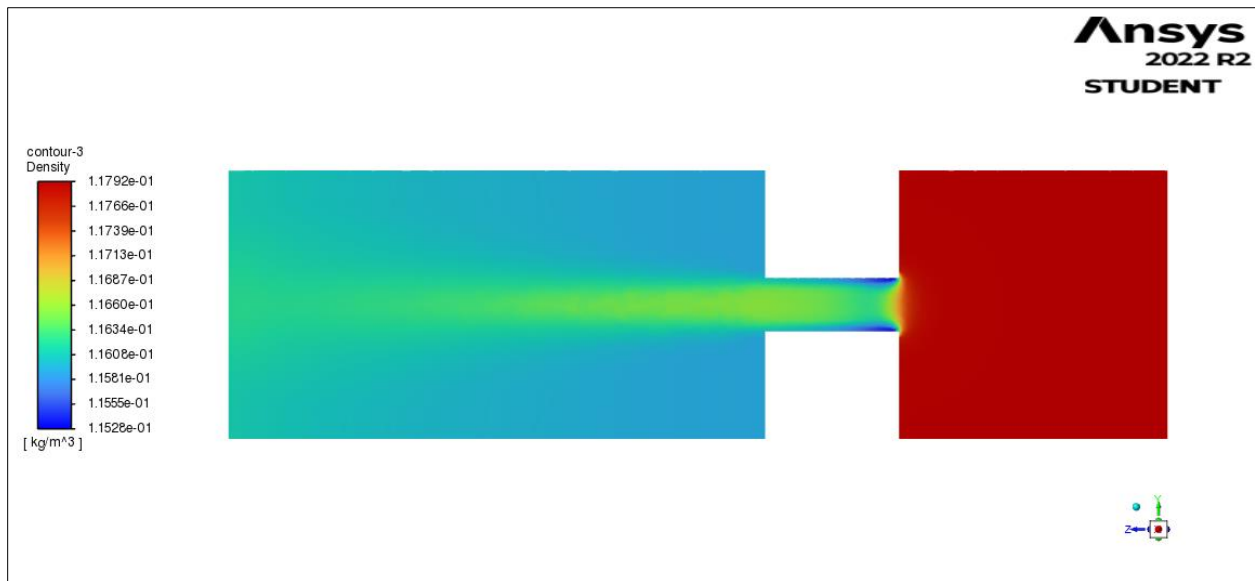


Figure 19: Contour plot of Density along symmetric plane YZ at 0.025 seconds of transient simulation.

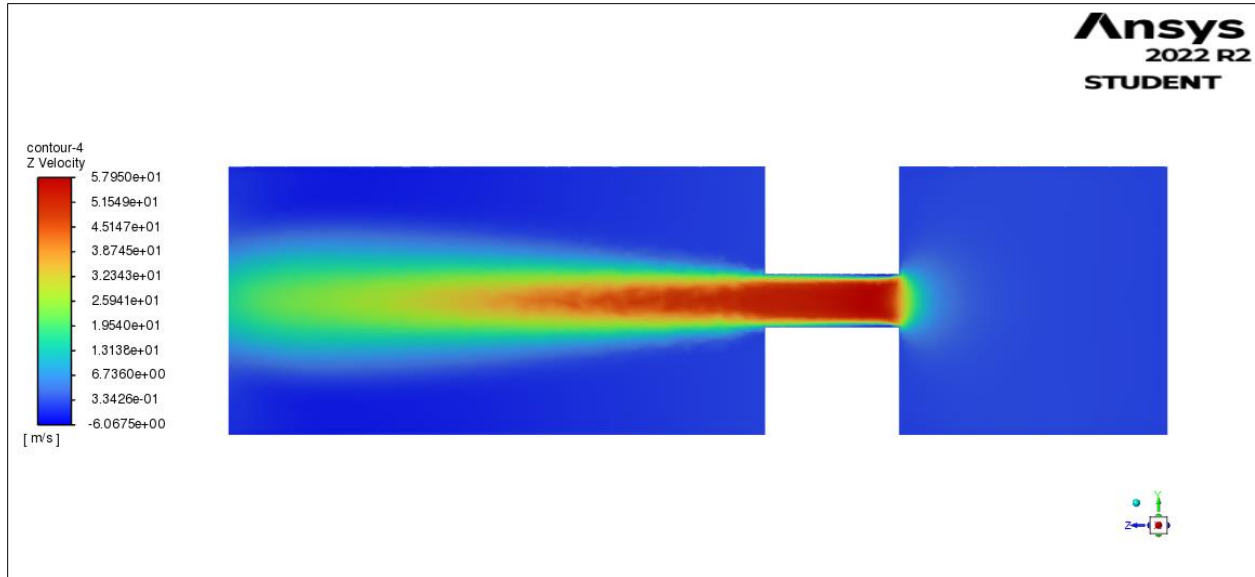


Figure 20: Contour plot of Z velocity component along symmetric plane YZ at 0.025 seconds of transient simulation.

The contour graph for static temperature at 0.05 secs shows that the temperature of chamber A has dropped further since more pressure is released from chamber A. But the temperature in major portion of chamber B is set as 300 K and hence the red color. A similar trend for density contour is seen at 0.05 secs. Velocity plot is like the one we found at 0.025 secs with slight variations in the extent of spread and lesser velocity magnitudes as can be expected.

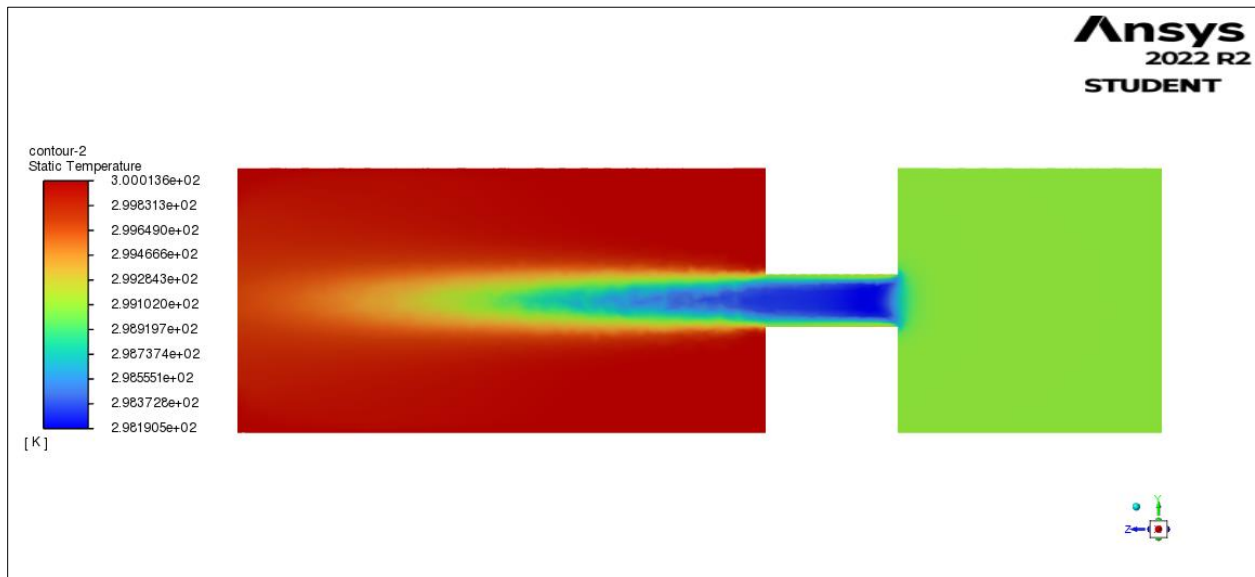


Figure 21: Contour plot of Static temperature along symmetric plane YZ at 0.05 seconds of transient simulation.

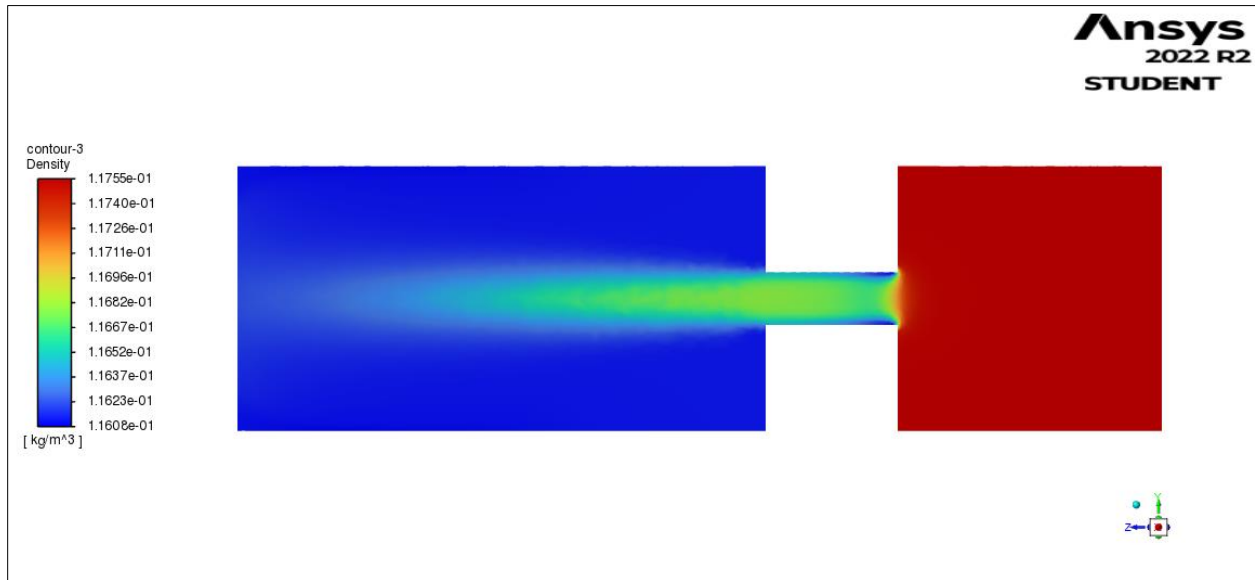


Figure 22: Contour plot of Density along symmetric plane YZ at 0.05 seconds of transient simulation.

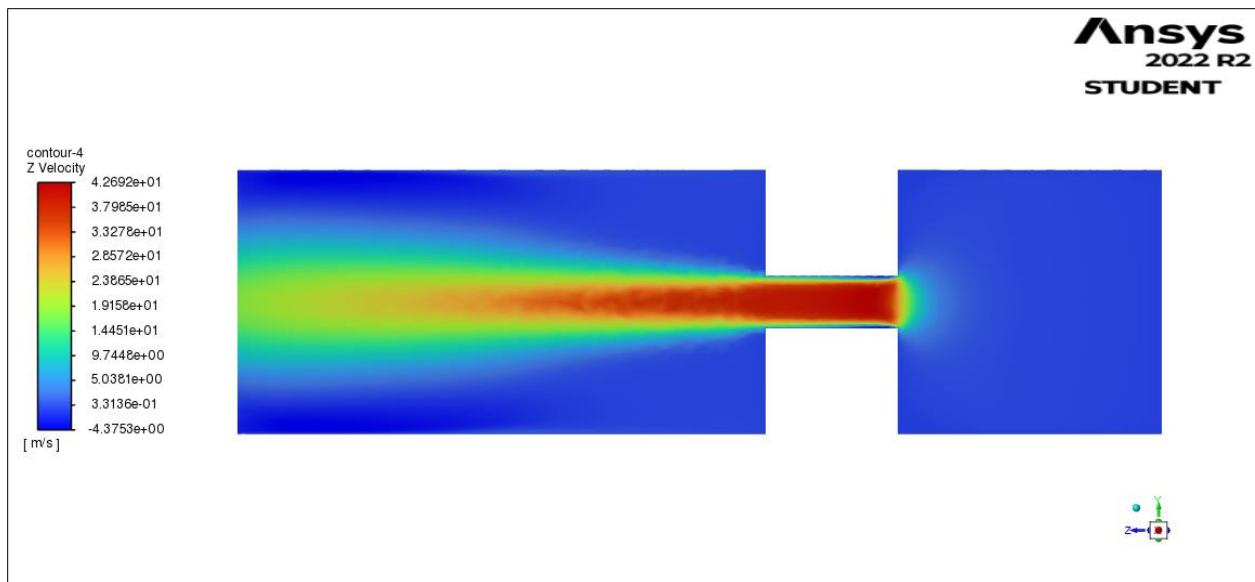


Figure 23: Contour plot of Z velocity component along symmetric plane YZ at 0.05 seconds of transient simulation.