

Initial setup –

I referred tutorial 1 given by Prof. Huang for initial setup except following changes –

1. I turned gravity “on” and put -9.81 in Y axis (along height 1 m of cylindrical tank)
2. I created new material for water and gave following values
 - a. Density - "Boussinesq" to allow buoyancy-driven thermal convection
 - b. Operating density = 986 kg/m³
 - c. Thermal expansion coefficient = $4 \times 10^{-4} \text{ 1/K}$
 - d. Operating temperature = 45 C
3. Set the boundary condition for the outlet to "outflow", instead of "pressure outlet".
4. Chose second order discretization
5. Used doubled precision for more accurate results
6. Refined mesh to $9 \times 10^{-3} \text{ m}$ with number of nodes almost equal to 48,000
7. Used k-epsilon model with “realization” mode on.
8. Used “Adaptive Gradient” for accurate results.

Task 1

$$T_{\text{out}} = \frac{\int \int v_n T dA}{\int \int v_n dA}$$

Steps to calculate Temperature

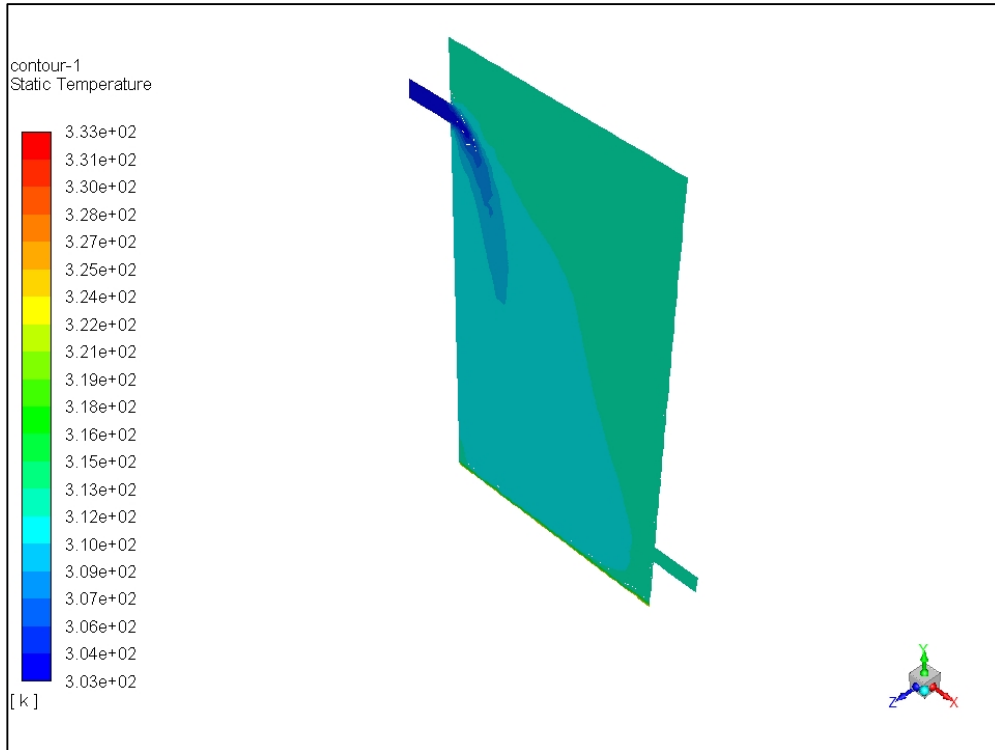
1. Create two different custom defined function for numerator and denominator
2. Use “integral” in “user defined” functions in “Results” to calculate the value of user defined function.
3. After getting the values, divide numerator by denominator and that is the value for Temperature.

For task 1

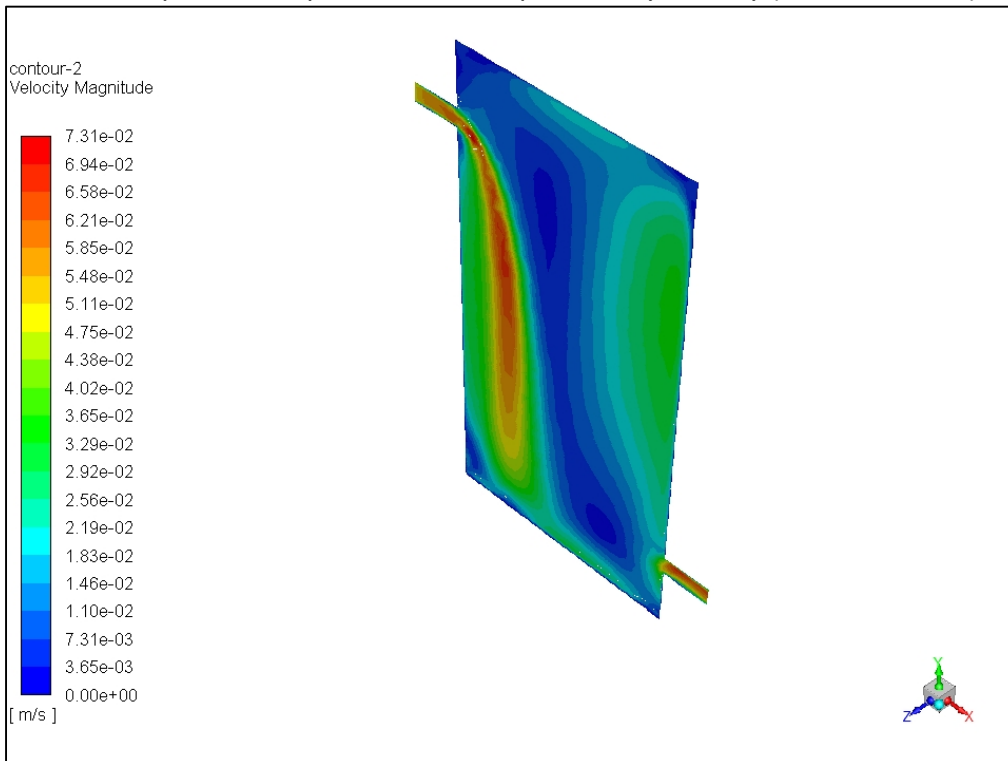
$$\int \int v_n T dA = 0.00874708$$

$$\int \int v_n dA = 2.7965489 \times 10^{-5}$$

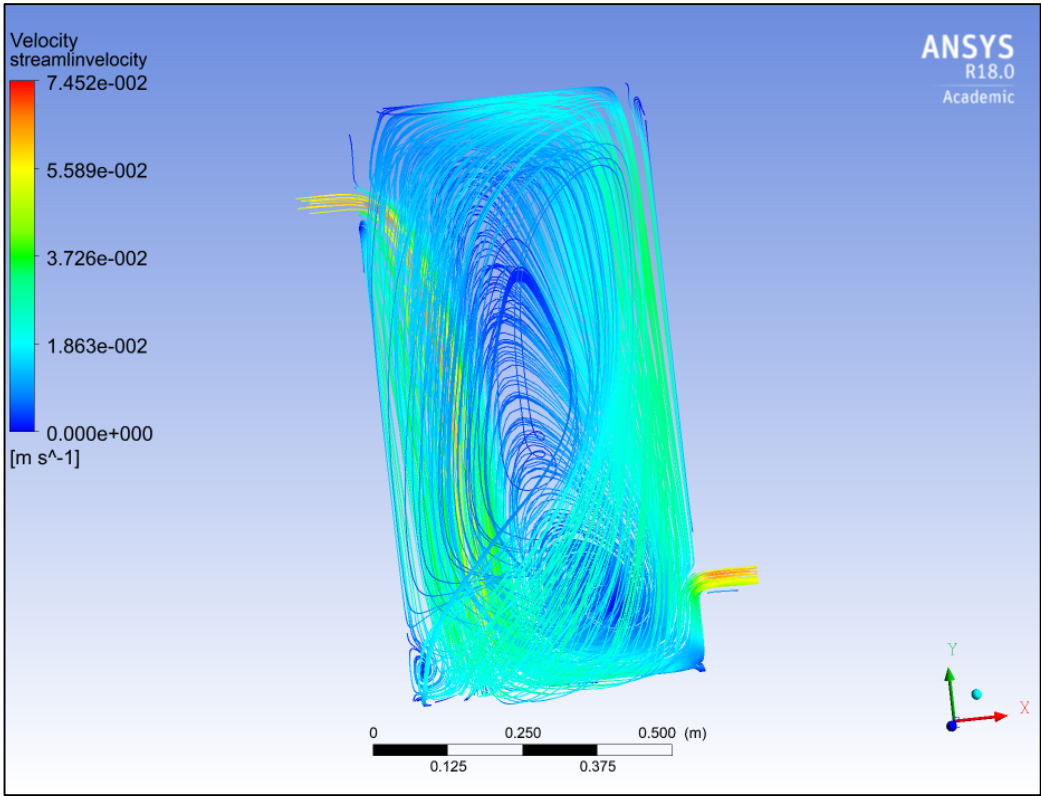
$$T_{\text{out}} = 312.7812145 \text{ K}$$



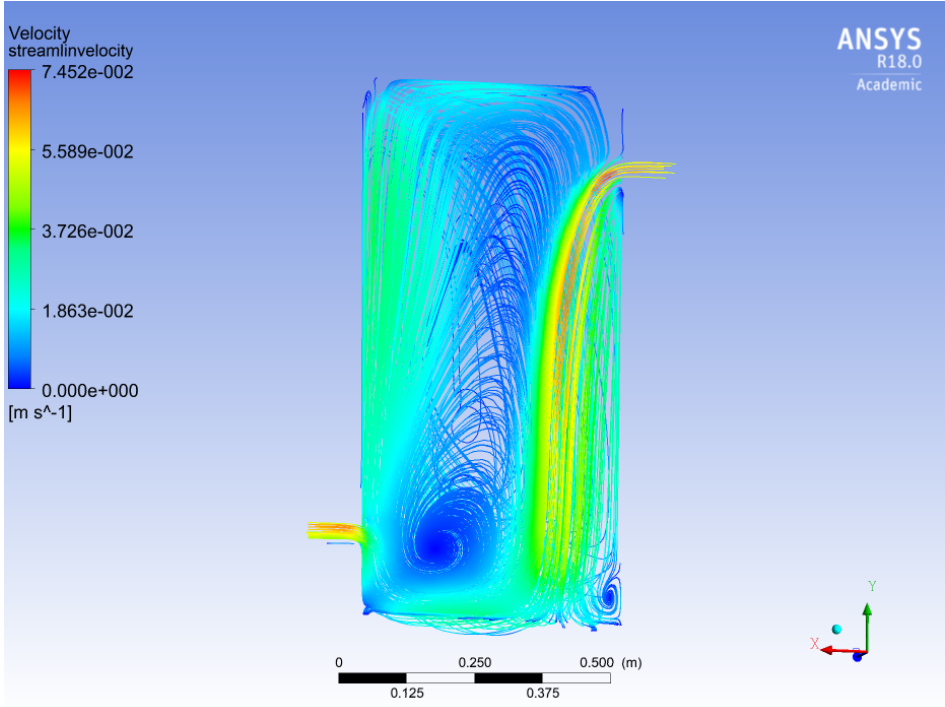
Contour plot of temperature on the plane of symmetry (isometric view)



Contour plot of velocity magnitude on the plane of symmetry (isometric view)



Plot of stream lines (front view)



Plot of stream lines (rear view)

Task 2

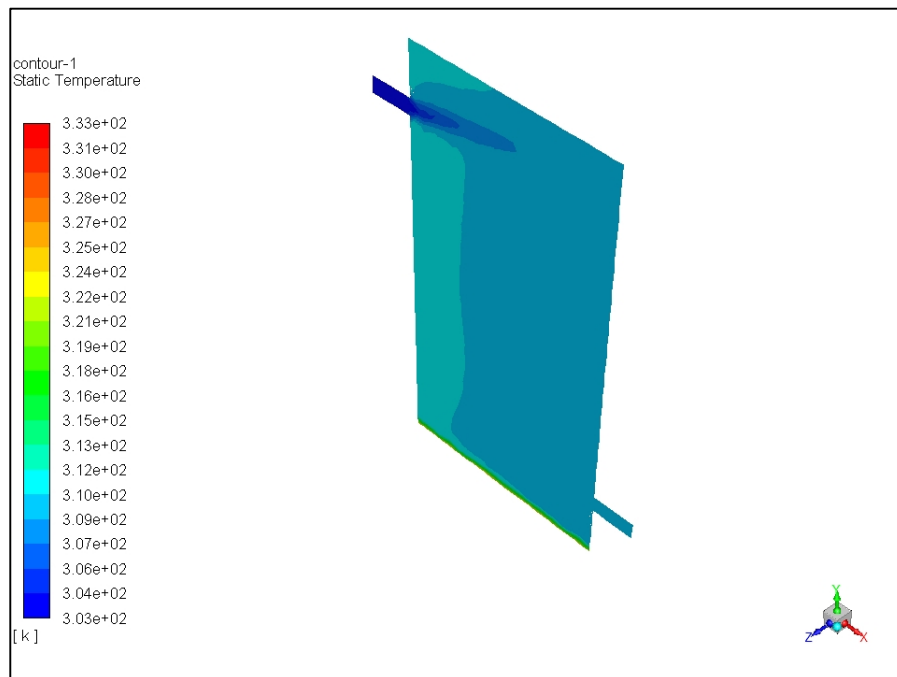
I changed direction of gravity from Y axis to X axis with the same value of 9.81 m/s². All other settings are same.

For task 2, using same steps as Task1 to calculate temperature, we get -

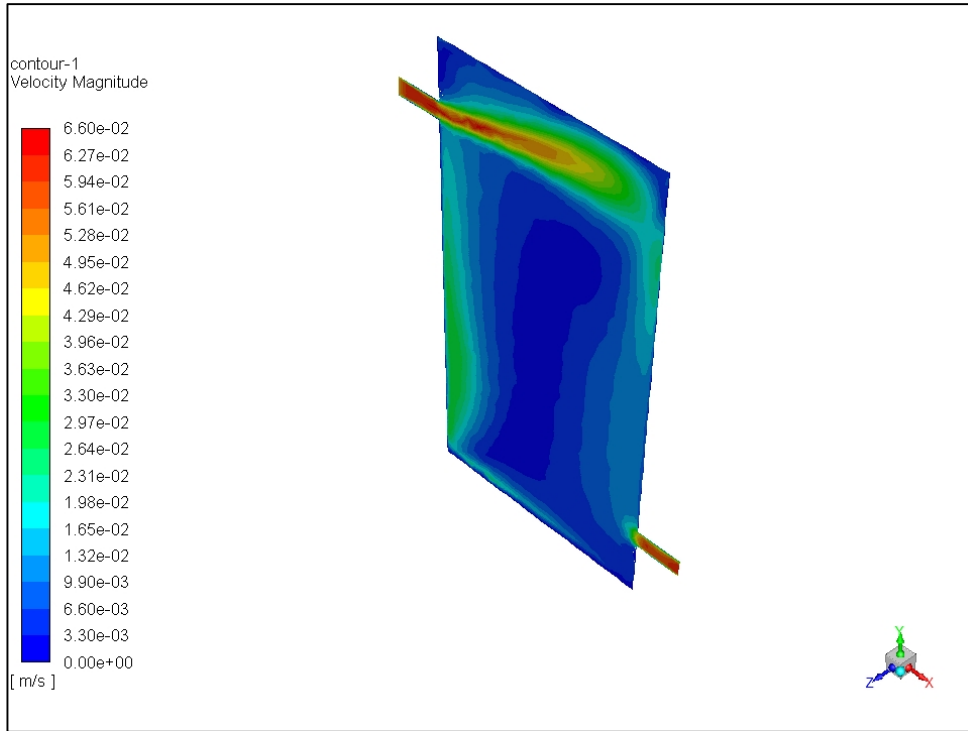
$$\int \int v n T d A = 0.0095174448$$

$$\int \int v n d A = 3.0713205 * 10^{-5}$$

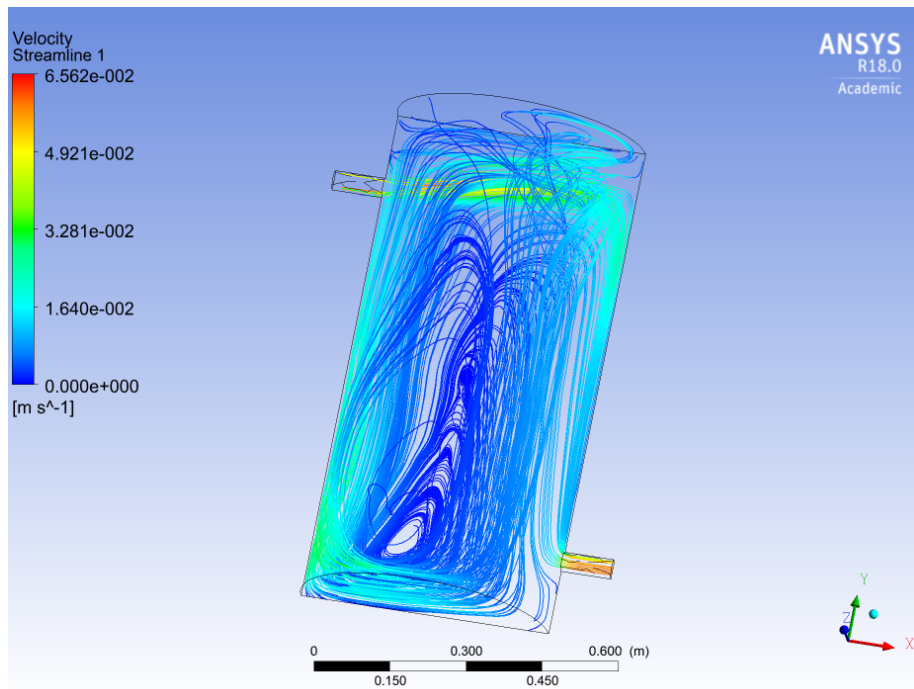
$$T_{out} = 309.881199 \text{ K}$$



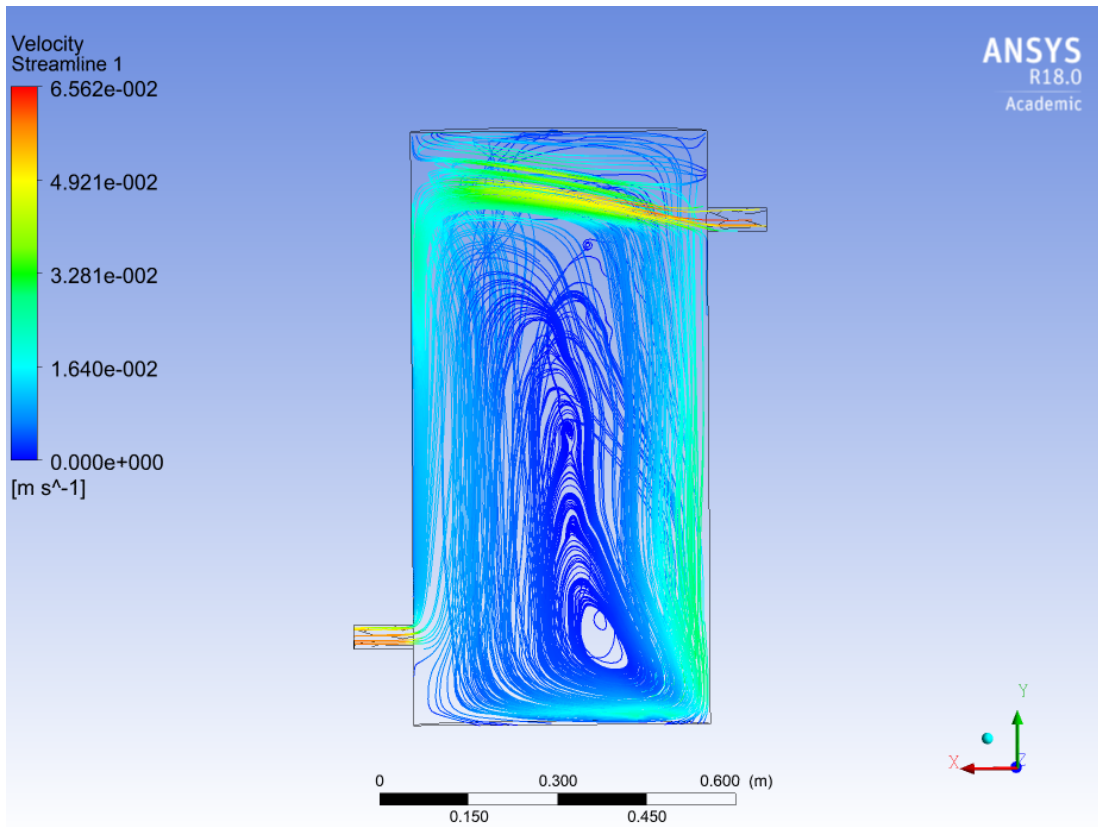
Contour plot of temperature on the plane of symmetry (isometric view)



Contour plot of velocity magnitude on the plane of symmetry (isometric view)



Plot of stream lines



Plot of stream lines (rear view)

Task 3

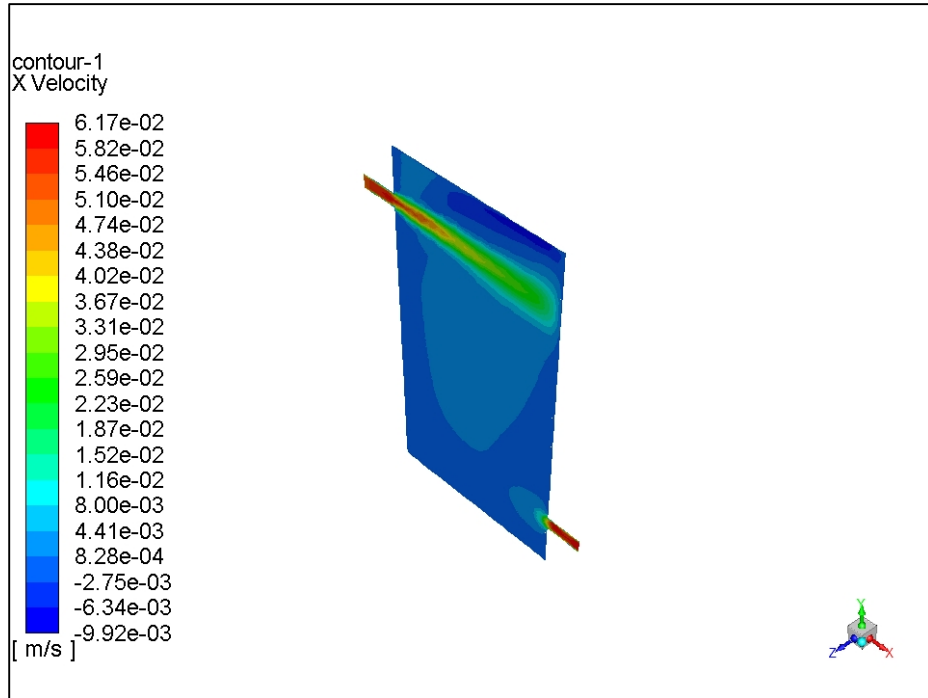
I turned gravity "off" for this task and density constant. All other settings are same.

For task 3, using same steps as Task1 to calculate temperature, we get -

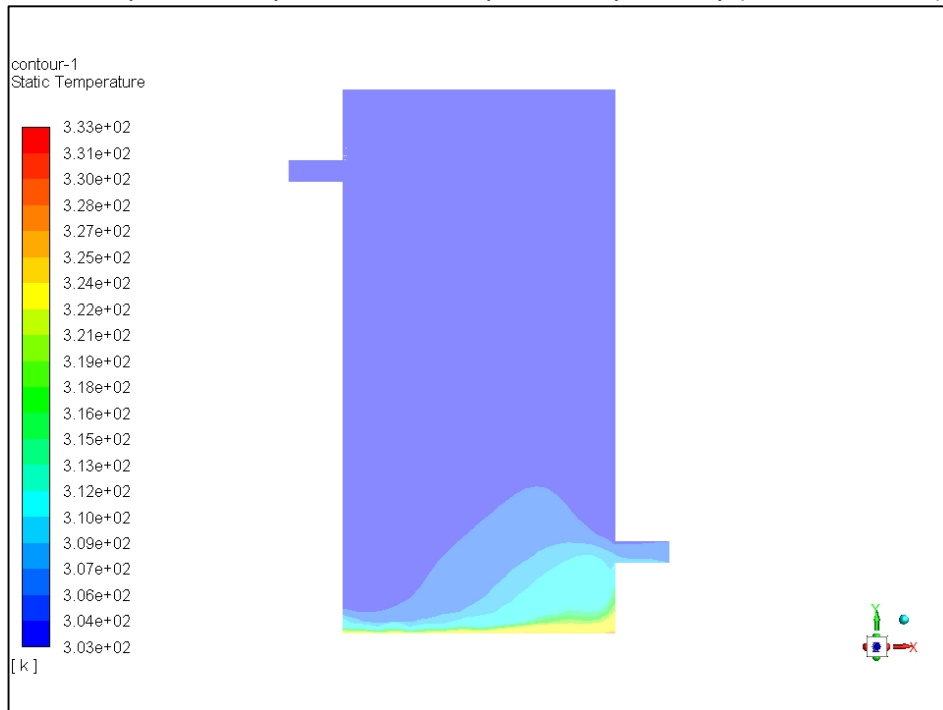
$$\iint v_n T dA = 0.0085905139$$

$$\iint v_n dA = 2.8151461 * 10^{-5}$$

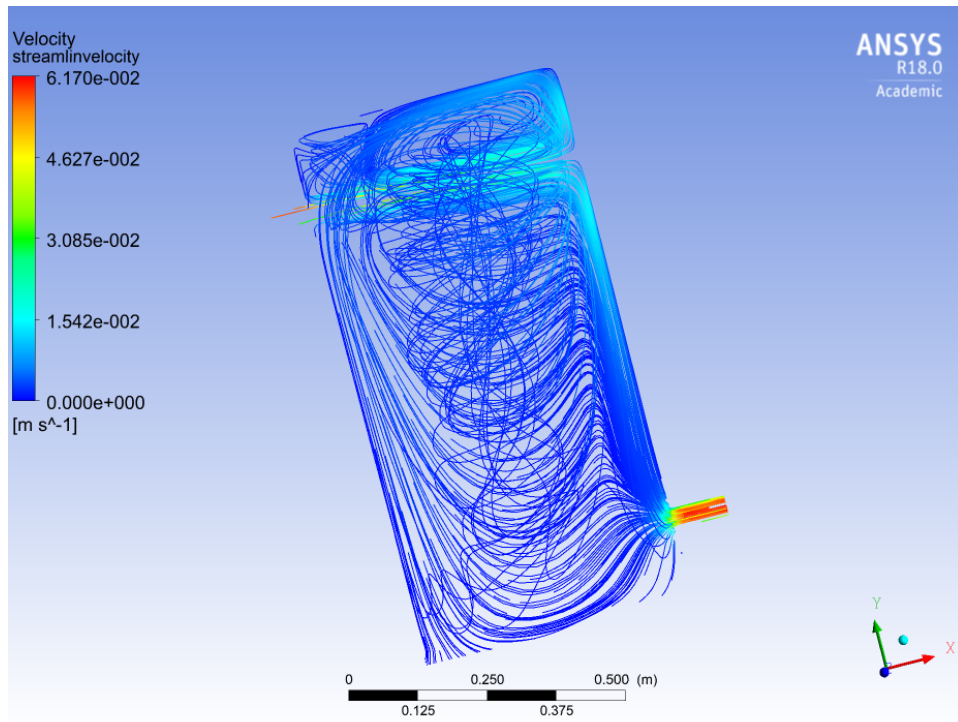
$$T_{out} = 305.15339 \text{ K}$$



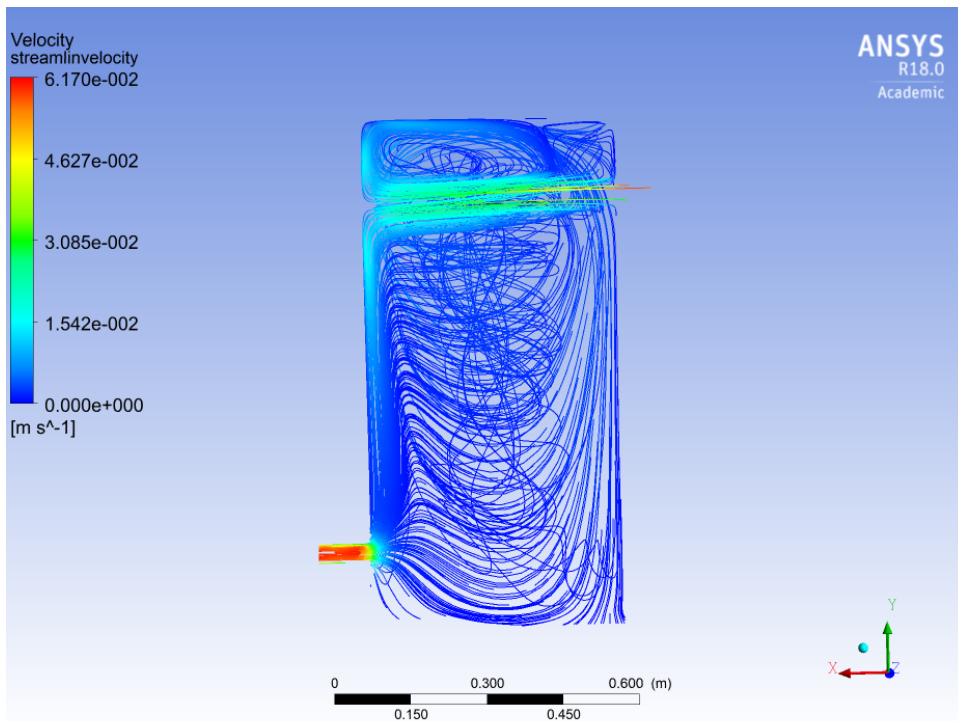
Contour plot of temperature on the plane of symmetry (isometric view)



Contour plot of velocity magnitude on the plane of symmetry (front view)



Plot of stream lines



Plot of stream lines (rear view)

Comparison of Task 1, 2 and 3 –

In task 1, the gravitational force is along the height of cylinder, i.e. normal to plate, hence the fluid is touching the surface, filling tank for some height, gaining heat and then leaving the tank, hence the outlet temperature is maximum for task 1 as compared to other two.

In task 2, the gravitational force is along X axis, hence, the fluid is moving towards the X axis (along the length of inlet/outlet pipe). Due to the gravitational force direction, not being normal to base plate, the amount of fluid taking heat from the plate will be less compared to task 1, hence, the outlet temperature is less than task 1.

In task 3, the only force is liquid's kinetic energy, due to which, the liquid will follow direction of velocity, collide from opposite wall, and due to collision, it will disperse and reach outlet. As there is no external force present, the amount of fluid reaching bottom will be very less, hence the outlet temperature is least in Task 3.

Task 4

For task 4, we had part a) initialization – 30 deg C b) initialization – 40 deg C.

1. Changed setting to “transient”.
2. Initialize temperature to “303 K” and “313 K” for part a) and b) respectively.
3. Define customized function for numerator and denominator as discussed in Task1
4. Time step used for this task was 1 sec
5. Mesh settings used – fine mesh
6. Total time steps for which solutions ran – 6604
7. The plots were then plotted using excel from the generated output files
8. All other settings remained same.

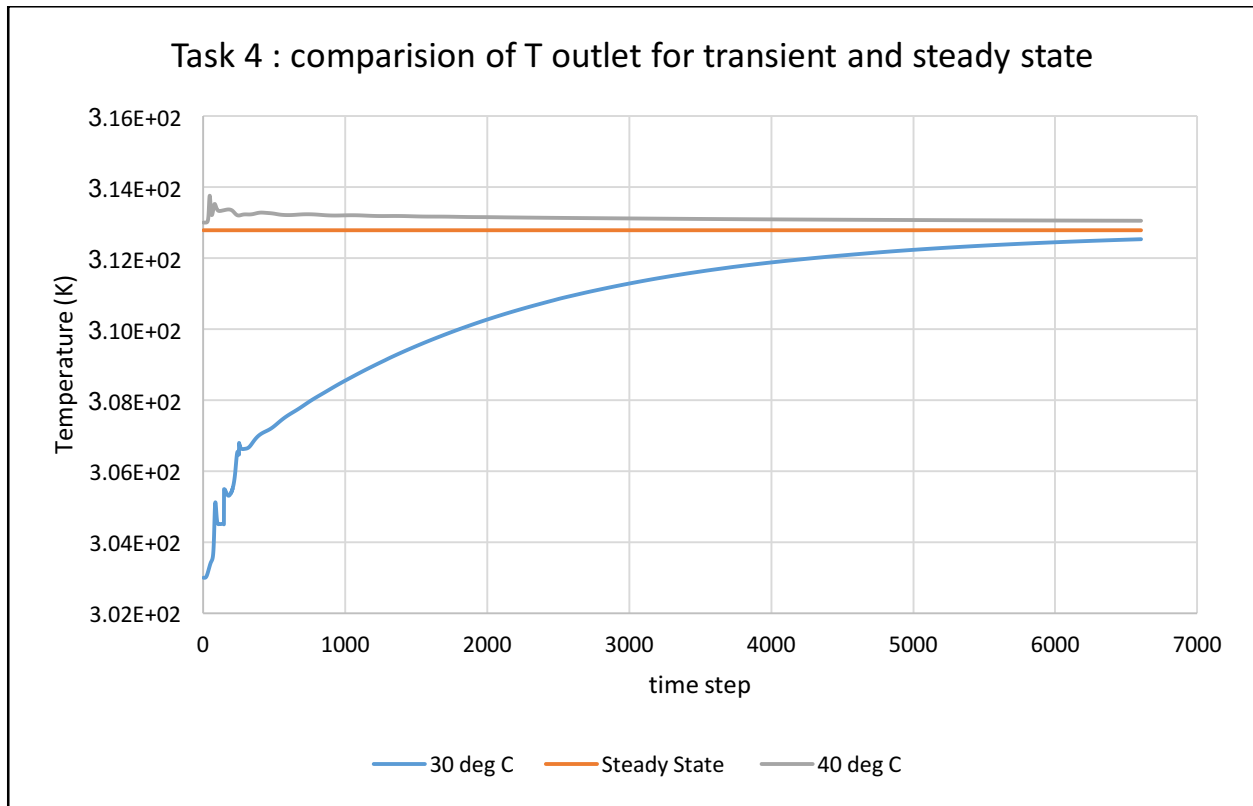
Steady state temperature = 312.78 K

For part a) Tout at last time step = 312.53 K

Temperature difference for part a) = 0.15 K < 0.3 K

For part b) Tout at last time step = 313.05 K

Temperature difference for part b) = 0.27 K < 0.3 K



Task 5

For task 5, I used the "Flux Report" function in Ansys-Fluent to obtain the total rate of heat transfer over the bottom plate, and then divided the total rate of heat transfer by the area of the bottom plate to obtain the average heat flux (in J s-1m-2) at the bottom plate.

$$\text{Average Heat Flux} = 1157.381 / (3.14 * 0.25^2 * 0.5) = 11794.96 \text{ J s-1m-2}$$

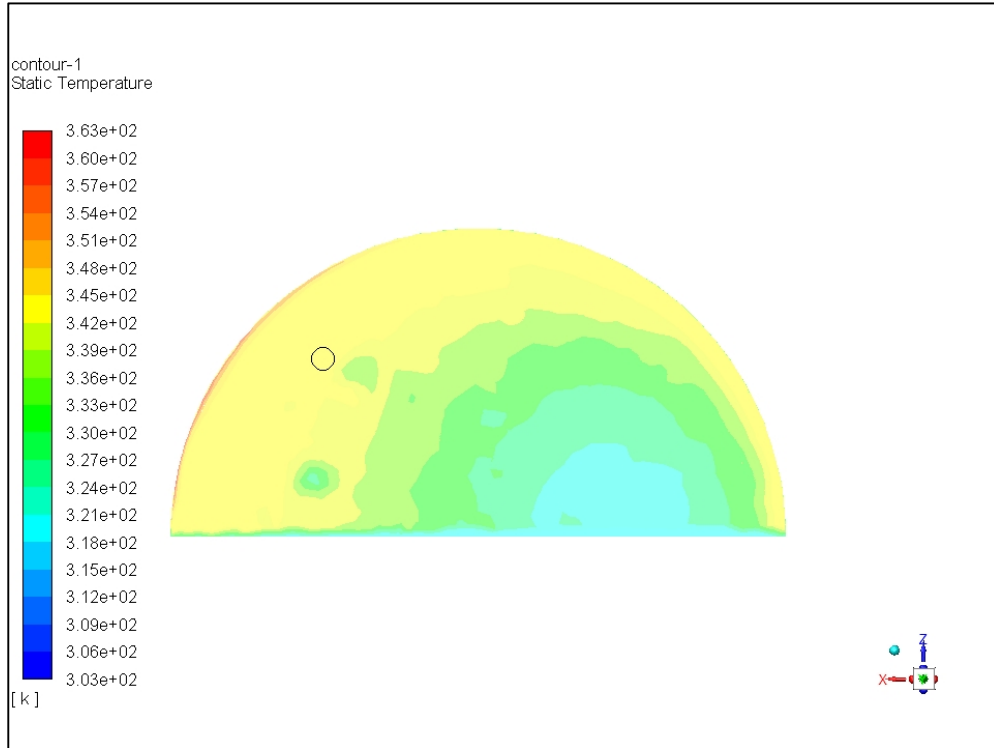
Using this value of heat flux instead of constant temperature, I ran simulation again (adaptive mesh), and then using same steps as Task1 to calculate temperature, we get -

$$\iint v n T d A = 0.0087582831$$

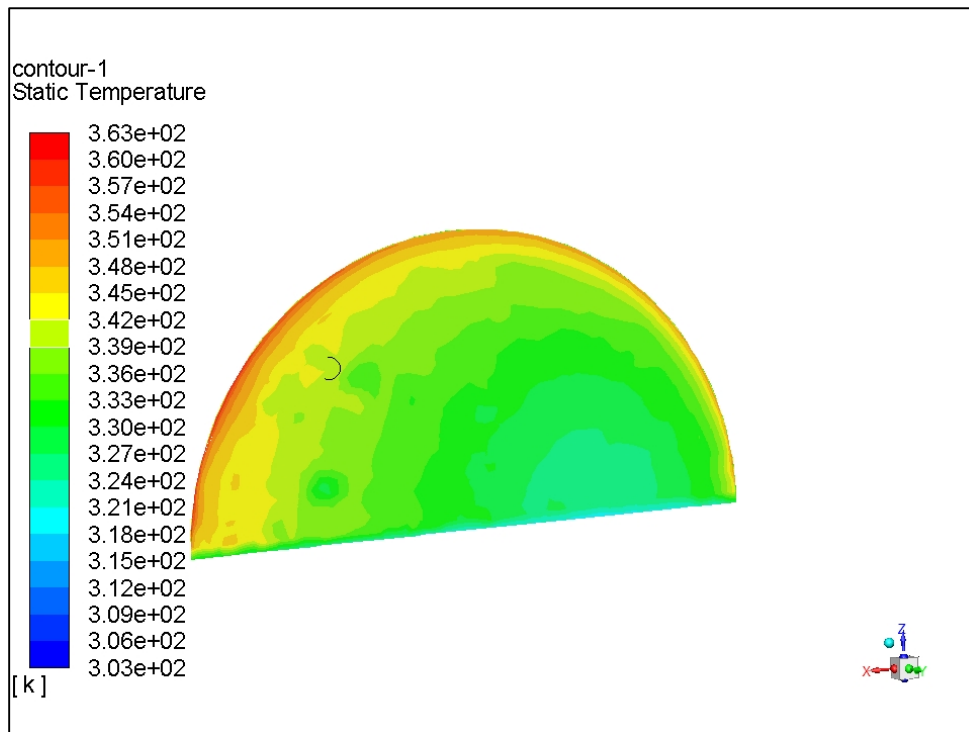
$$\iint v n d A = 2.7996258 * 10^{-5}$$

$$T_{out} = 312.83 \text{ K} \approx 312.78 \text{ K}$$

Outlet temperature for both cases are almost same. There is a slight difference because of conduction due to temperature difference.



contour plot of temperature on the bottom plate (bottom view)



contour plot of temperature on the bottom plate