

MAE 598 - Applied Computational Fluid Dynamics

Project-I

Submitted by:

Praveen Silori



SCHOOL FOR ENGINEERING OF MATTER, TRANSPORT & ENERGY

MECHANICAL ENGINEERING DEPARTMENT

Project 1

Note: No collaboration was done in doing the present Project-1.

Task 1

Part a

To simulate the fluid flow and heat transfer in a water heater, with water as a working fluid. The geometrical parameters are: diameter of the tank $D = 0.6$ m, height of the tank $H = 0.8$ m, diameter of the inlet and outlet side pipes $d = 0.06$ m and length is $L = 0.15$ m, location of the inlet and outlet side pipe from the bottom wall $Z_1 = 0.6$ m and $Z_2 = 0.2$ m respectively. The boundary conditions are: velocity inlet on the inlet side pipe with $V_{in} = 0.06$ m/s with temperature $T_{in} = 20^\circ$ C. The outlet side pipe is assigned out flow condition. The bottom wall of the heater is assigned a temperature boundary condition as $T = 55^\circ$ C. Standard k-epsilon turbulence model is used in the present simulation with full buoyancy effect. The pressure based solver is to be used in the present simulation and steady state solution is desired. The gravitational acceleration is taken as $g = -9.81$ m/s^2 .

The density is set as Boussinesq for water in order to allow density variation with temperature. The operating density is evaluated from below Eq. 1 from the given reference and its value is $\rho = 993.1448623$ kg/m^3 . Based on the density, the thermal expansion coefficient is evaluated from below Eq. 2 and its value is $\beta = 0.00036621$ $1/K$. In the given equation T is the operating temperature and its value is average of maximum and minimum of temperatures in the system which is $T = 37.5^\circ$ C.

$$\rho = 999.85308 + 6.3269 \times 10^{-2} T - 8.523829 \times 10^{-3} T^2 + 6.943248 \times 10^{-5} T^3 - 3.821216 \times 10^{-7} T^4 \quad (1)$$

$$\beta = -1/\rho (d\rho/dT) \quad (2)$$

Part b

To simulate the given problem with the reduced gravity $g = -3.72$ m/s^2 .

Solution

Task 1(a)

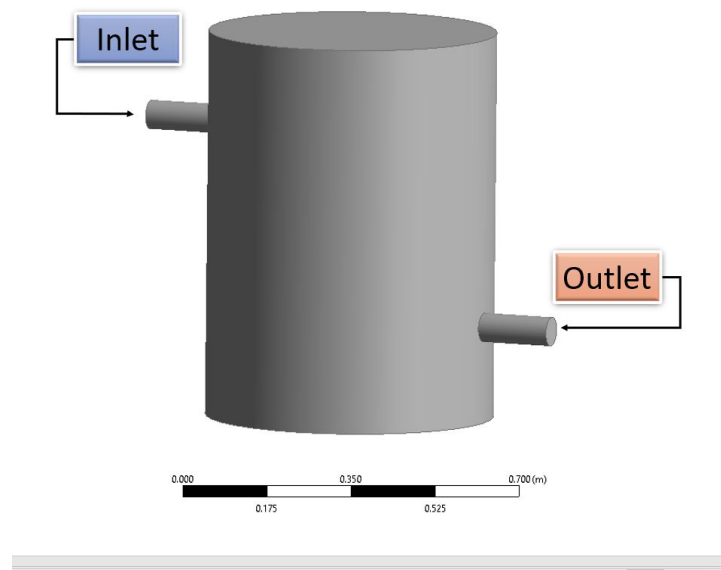
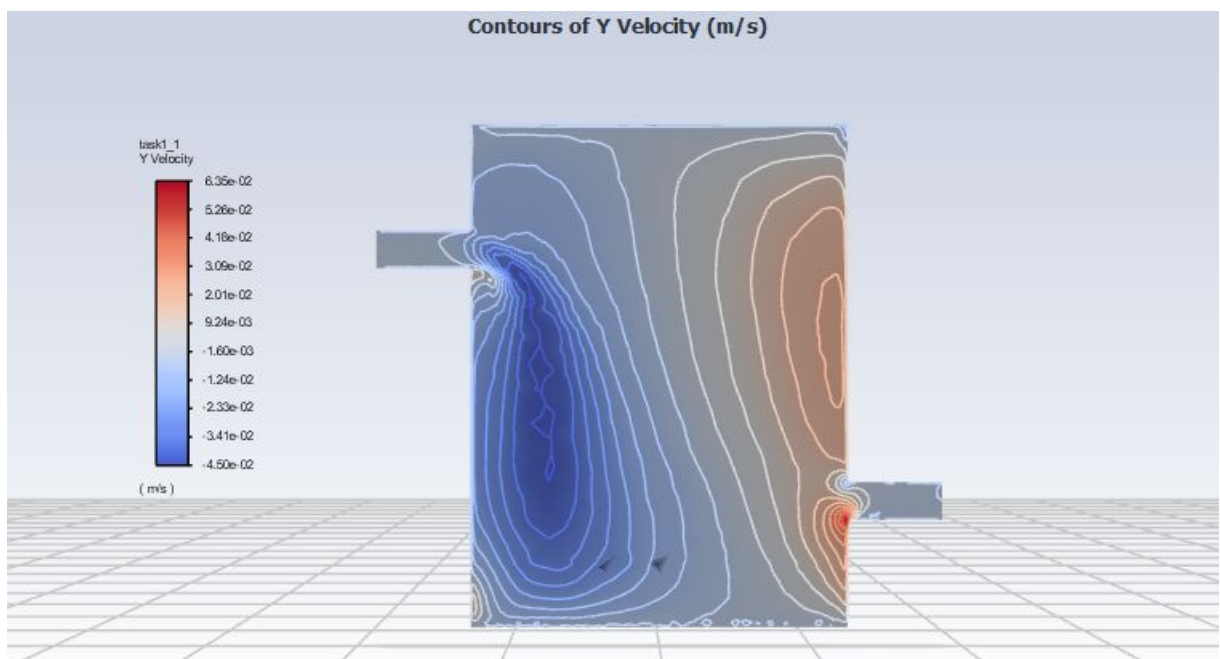


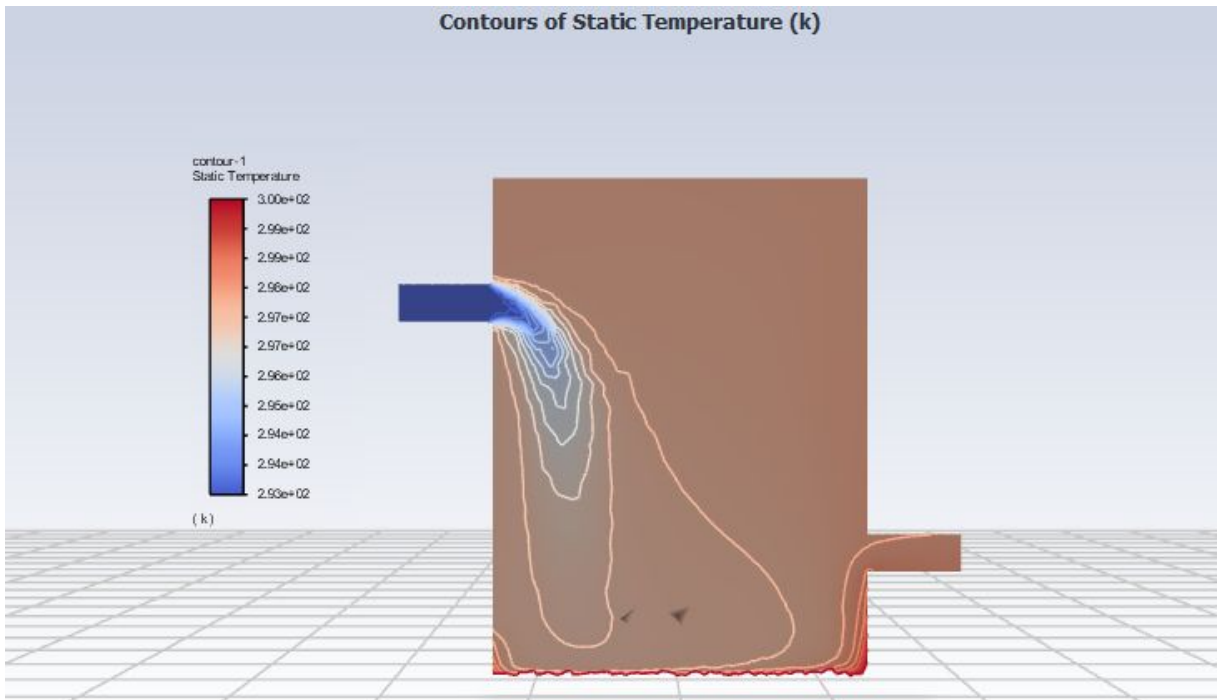
Figure 1 shows the computational domain of the water heater used in the present simulation.

Deliverable (1 & 2)

The velocity and the temperature contour plot is shown in Fig 2(a) and (b)



(a)



(b)

Figure 2: (a) Velocity contour at the plane of symmetry and (b) Temperature contour at the plane of symmetry.

Deliverable (3)

The outlet temperature at the steady state is $T = 298.01$ K which is calculated by the Equation (3)

$$T_{out} = \frac{1}{A} \iint_A T dA \quad (3)$$

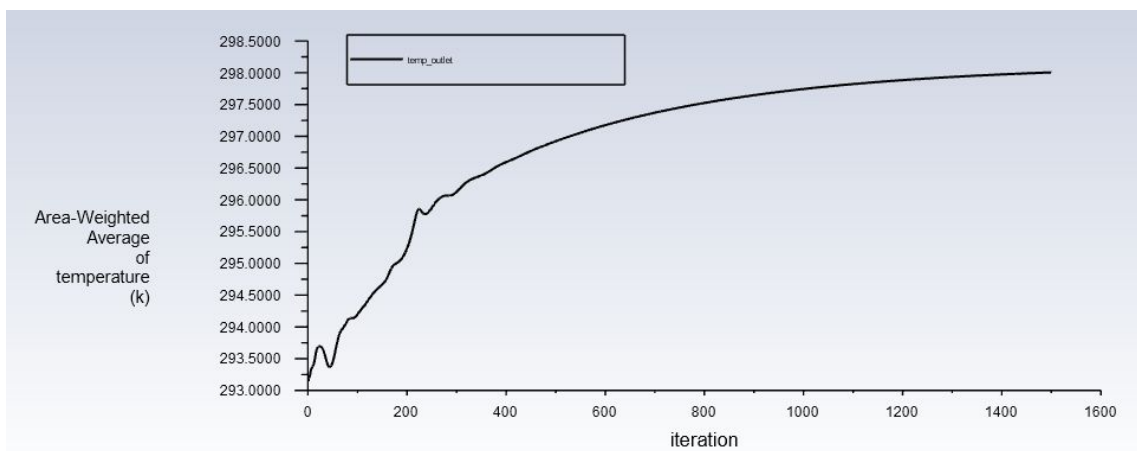


Figure 3: Line plot of the outlet temperature as a function of the number of iterations.

The convergence criteria is achieved when the temperature between two iterations with an interval of 100 is less than 0.1 K. The temperature difference is observed as 0.04 K over the 1400 and 1500 interval.

Deliverable 4:

Fig. 4 shows the mesh (on the plane of symmetry) that is used in this simulation. The element size for the triangular mesh is taken as 2.3 cm. The inflation is led by 'Program Controlled'.

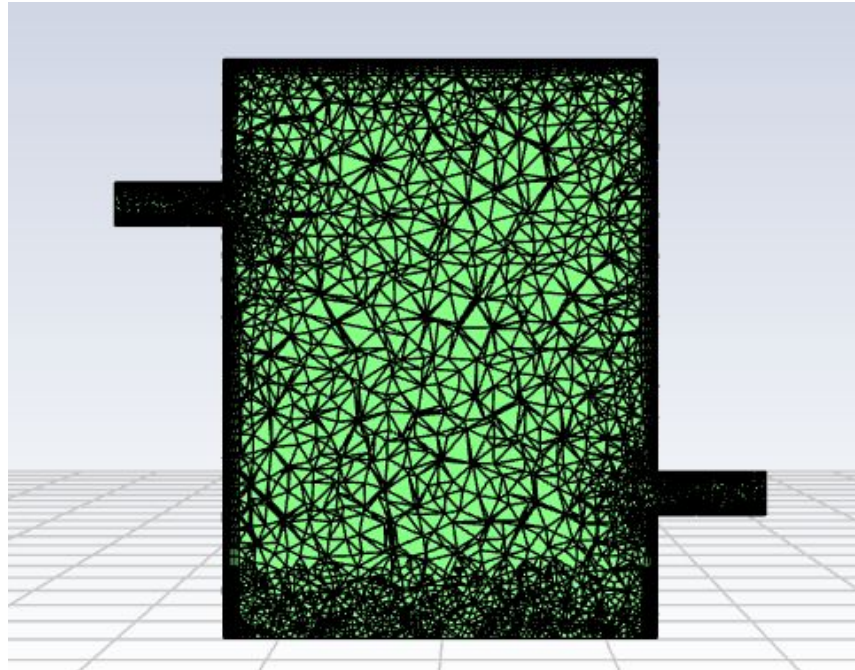
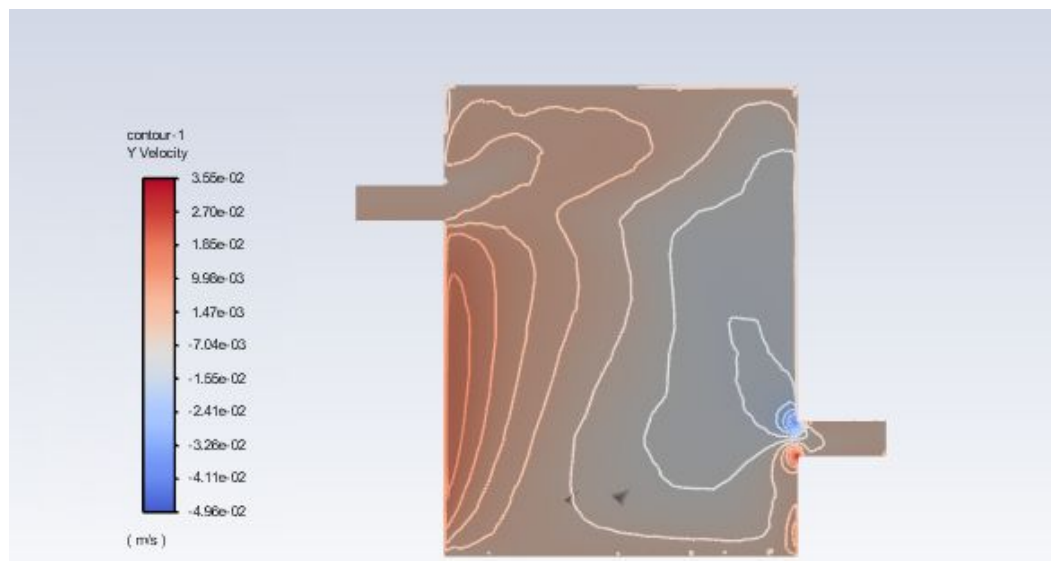


Figure 4: Mesh resolution at a plane of symmetry. Local mesh refinement at the bottom of the cylinder.

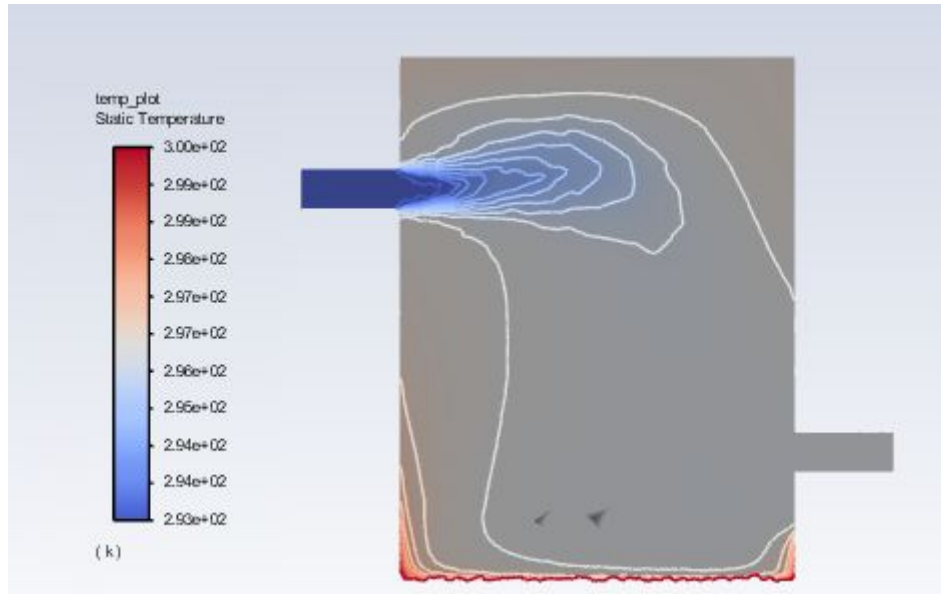
Task 1(b)

Deliverable (1 & 2)

The velocity and the temperature contour plot is shown in Fig. 5 when $g = -3.72 \text{ m/s}^2$.



(a)



(b)

Figure 5: (a) Velocity contour at the plane of symmetry and (b) Temperature contour at the plane of symmetry.

Deliverable (3)

The outlet temperature at the steady state is $T = 296.33$ K which is calculated using Eq. 3. The convergence criteria is achieved when the temperature between two iterations with intervals of 100 is less than 0.1 K. The temperature difference is observed as 0.06 K over the 1400 and 1500 interval.

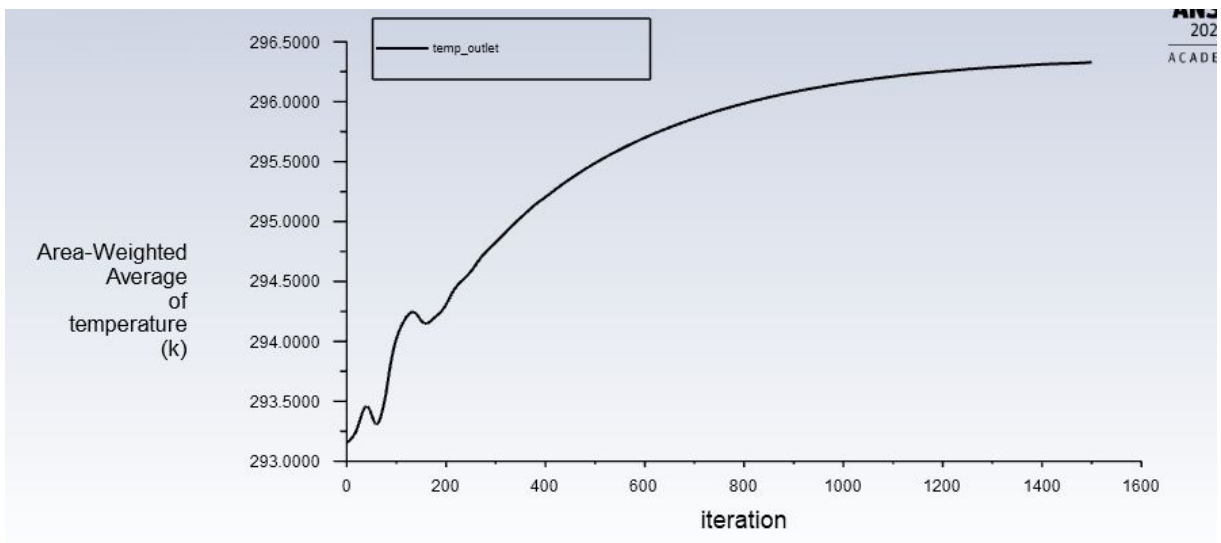


Figure 6: Line plot of the outlet temperature as a function of the number of iterations.

Task 2

To simulate the Fluid flow and heat transfer in a coiled water heater, with water as a working fluid. The helical pipe with its center traced by the equation of the helical curve as follows: $X(t) = R*\cos(t)$, $Y(t) = R*\sin(t)$, and $Z(t) = C*t$. The curve is traced from $t = 0$ to $t = 12\pi$ with $R = 0.3$ m and $C = 0.15/2\pi$ m. The helical pipe has circular cross section with $d = 8$ cm. The boundary conditions are: velocity inlet at inlet with four different values $V_{in} = 0.01, 0.02, 0.04, \text{ and } 0.08$ (m/s) respectively, with temperature $T_{in} = 300$ K for all the cases. The outlet is assigned as an outflow condition. The wall of the heater is assigned heat flux boundary condition with $Q = 600$ W/m². Laminar model to be used in the present simulation. Pressure based solver is to be used in the present simulation and steady state solution is desired. No gravity to be considered in the present simulation and density is set to be constant for water.

Solution

Figure 7 (a) shows the computational domain of the coiled water heater and Fig. 7(b) shows the meshed model with a size of 1.2 cm and the inflation is led by 'Program Controlled'.

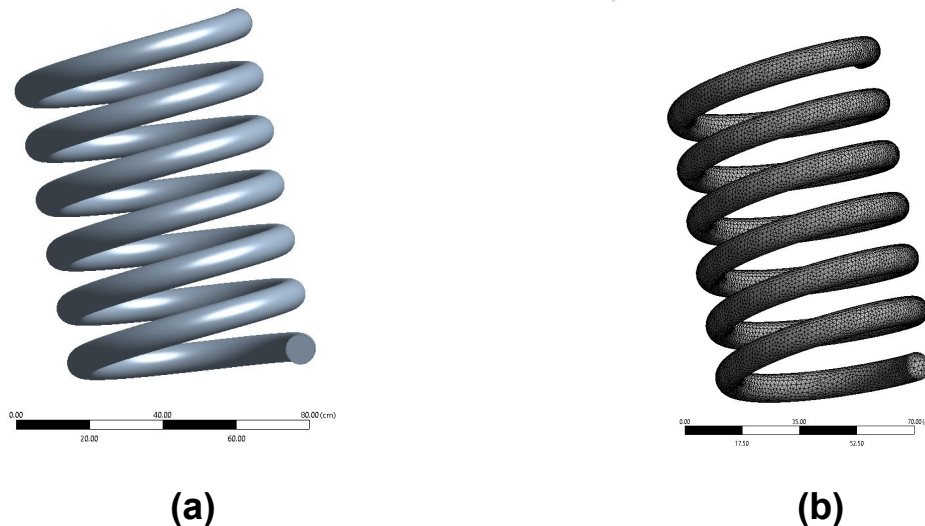


Figure 7: (a) Computational domain of coiled water heater, (b) meshed model generated

Deliverable (1)

The values for $\Delta T = T_{out} - T_{in}$ for the four different inlet velocities are given in Table 1. Figure 8 shows the plot for ΔT v/s V_{in} where ΔT is the numerical value obtained by using Eq. 3.

T_{out} (K)	$\Delta T = T_{out} - T_{in}$ (K)	V_{in} (m/s)
308.69	8.6900	0.01
304.43	4.4300	0.02
302.25	2.2500	0.04
301.13	1.1300	0.08

Table 1: Values for ΔT for different inlet velocities V_{in}

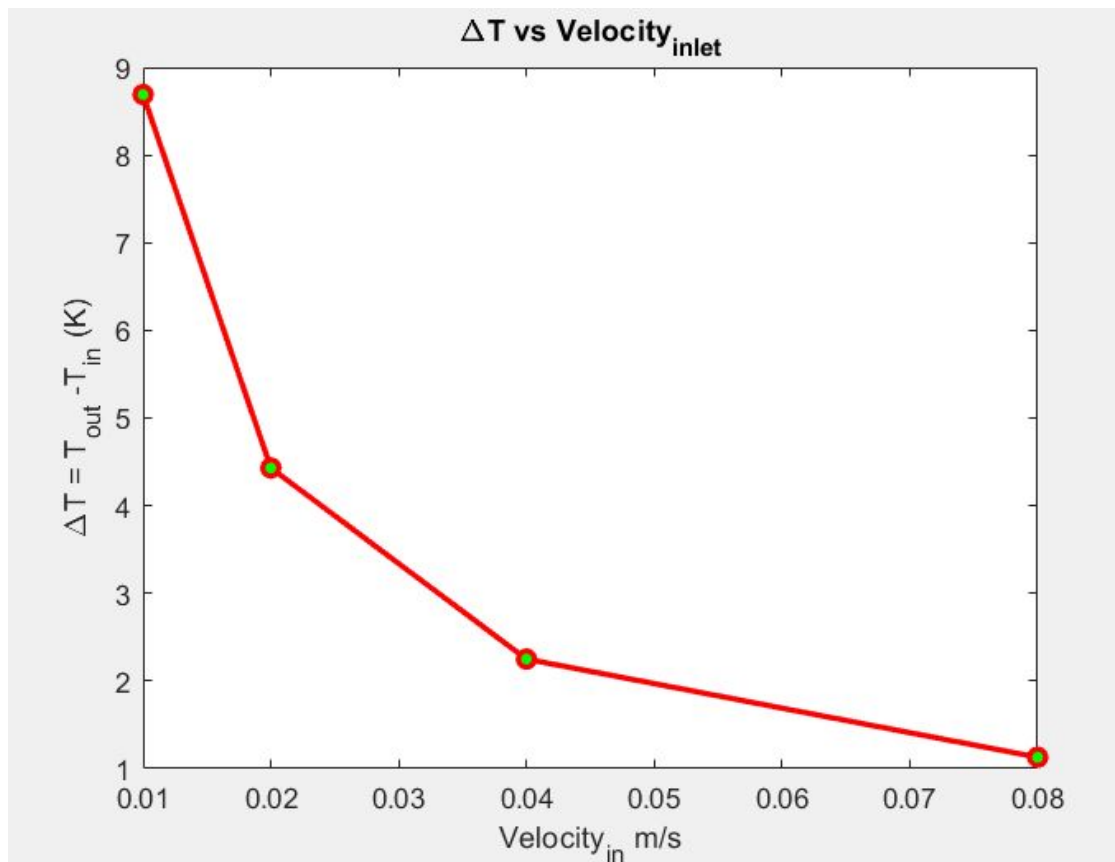
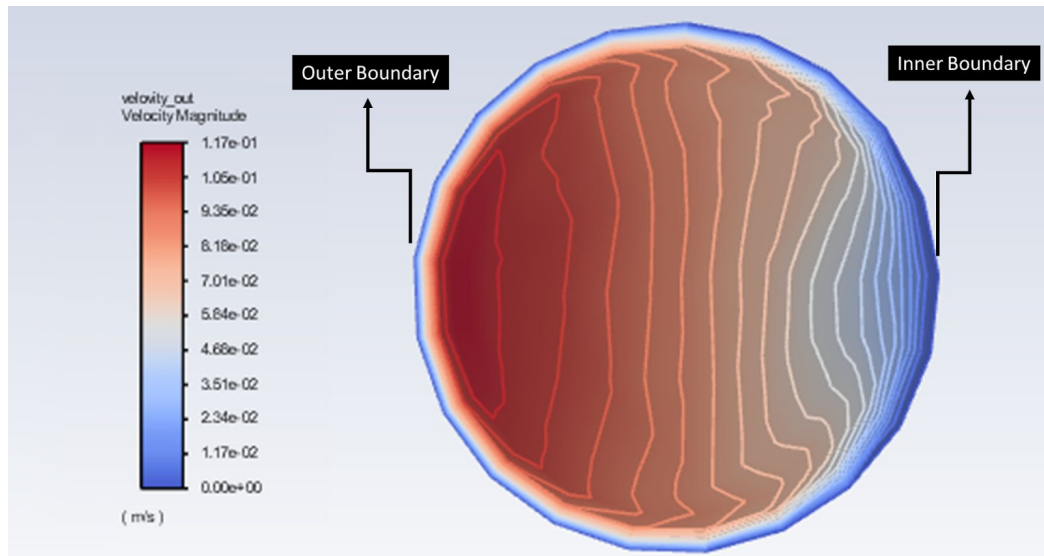


Figure 8: Plot for $\Delta T_{Numerical} = T_{out} - T_{in}$ v/s the inlet velocity V_{in}

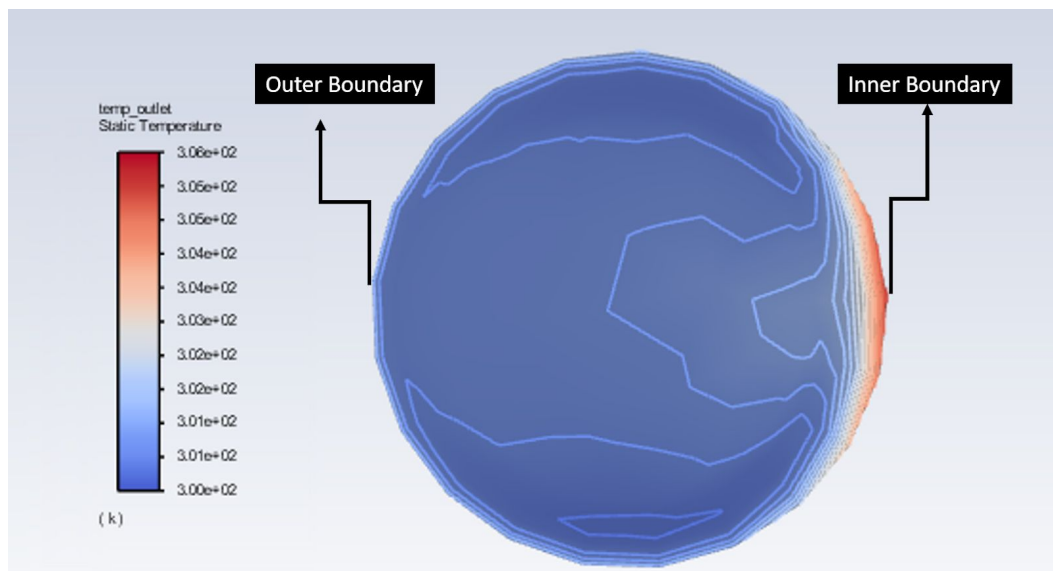
This graph shows that the velocity at inlet is inversely proportional to the temperature difference. The same can be deduced through analytical calculations. The reason behind this fact is that as the fluid velocity increases in the pipe, the residence time decreases that affects the temperature rise and as result the outlet temperature is less. The same is shown in Figure 8.

Deliverable (2)

The contour plot for the velocity magnitude and temperature at the outlet surface for the case of $V_{in} = 0.08$ (m/s) is shown in Fig. 9.



(a)



(b)

Figure 9: (a) Velocity magnitude at the outlet surface and (b) Temperature contour at the outlet surface.

As observed from Fig. 9 (a), the water velocity at the outer edge of the pipe is higher as compared to the water velocity at the inner edge. This is mainly due to the fact that water particles nearer to the outer edge have to travel more distance in the same interval of time as compared to those which travel nearer to the inner edge. Due to this phenomena, the heat gained by water nearer to the outer edge will be less as compared to heat gained by water nearer to the inner edge. As a result, the temperature of water nearer to the outer edge will be less as compared to the

temperature of water nearer to the inner edge. The same can be concluded from Fig. 9 (b).

Task 3

To simulate the fluid flow and heat transfer in a chamber, with air as a working fluid. The geometrical parameters are: diameter of the chamber $D = 30$ cm, height of the chamber $H = 15$ cm, diameter of the outlet side pipe $d = 5$ cm, length of the side pipe is 10 cm. The boundary conditions are: the outlet side pipe is assigned as pressure outlet condition with 0 gauge pressure and 20°C as back flow temperature. The bottom and top wall of the chamber is assigned as wall boundary condition with heat generation as 100 (W/m^2). The Laminar model is used in the present simulation. The density based solver is to be used in the present simulation and transient simulation is to be performed. The density to be set as 'Ideal Gas' for air.

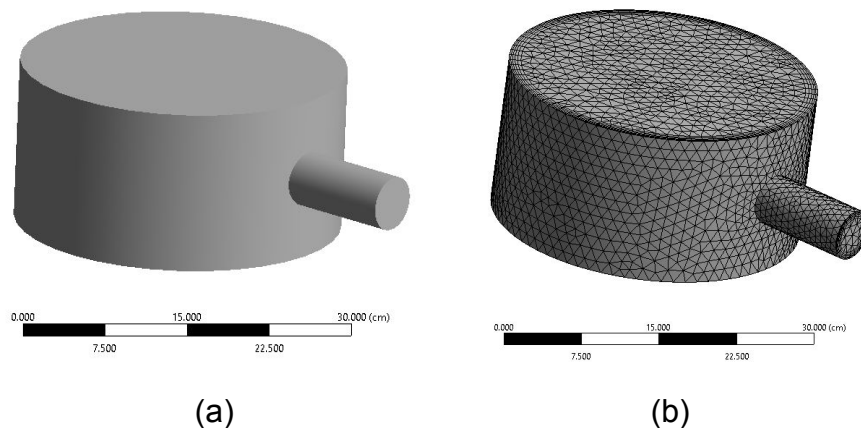
To calculate the area weighted average velocity, V_{out} , normal to the surface of the outlet based on Eq. 4

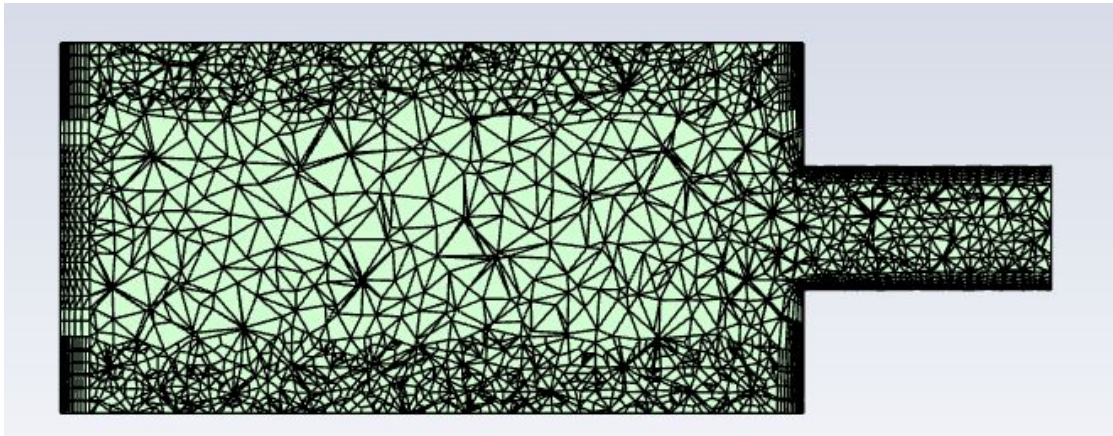
$$V_{out} \equiv \frac{1}{A} \iint_A v \, dA \quad (4)$$

where, v is the velocity component normal to the surface of the outlet.

Solution

Figure 10 (a) shows the computational domain of the chamber and Fig. 10 (b) shows the grid used at the plane of symmetry in the present simulation. In the present simulation the element size for grid generation is taken as 1 cm. Program controlled inflation is allowed at the boundary of the chamber.





(c)

Figure 10: (a) Computational domain for the chamber, (b) Meshed Model (c) Local mesh refinement at top and bottom (shown at plane of symmetry).

Deliverable (1)

The line plot for V_{out} v/s time from $t = 0$ s to $t = 5$ s is shown in Fig. 11. As the temperature of the air increases in the chamber it expands and as a result a continuous increase in the velocity (area weighted average) of the air, normal to the outlet, is observed. In the present simulation, the time step size is chosen as $\Delta t = 0.1$ s. The number of time steps is kept as 50 to achieve the solution till $t = 5$ s. The maximum iterations per time step is chosen as 50. The value of V_{out} is 0.018 (m/s) at $t = 5$ s.

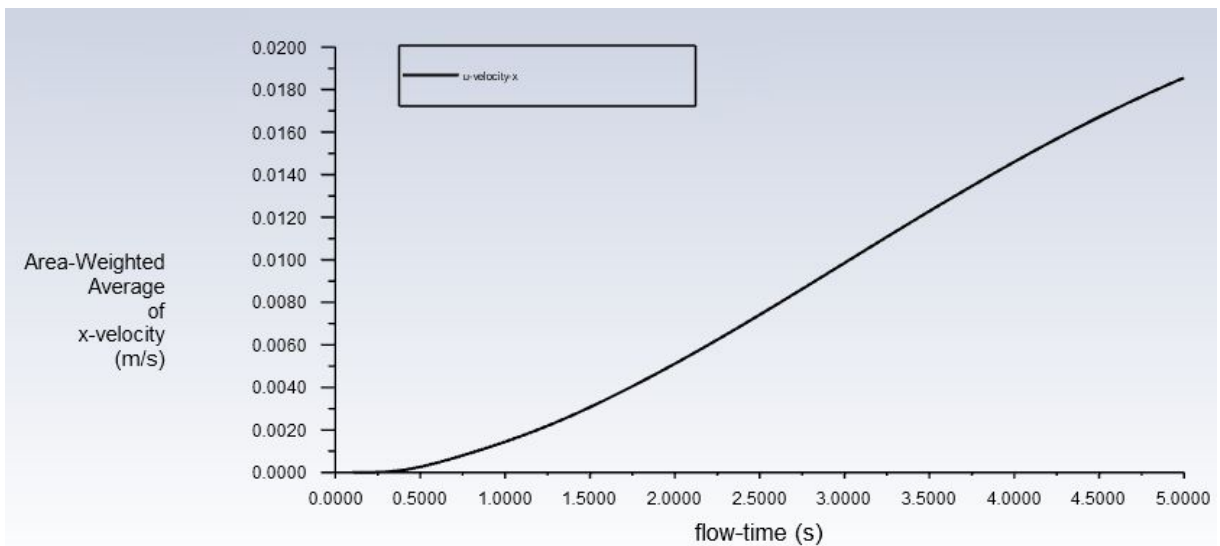
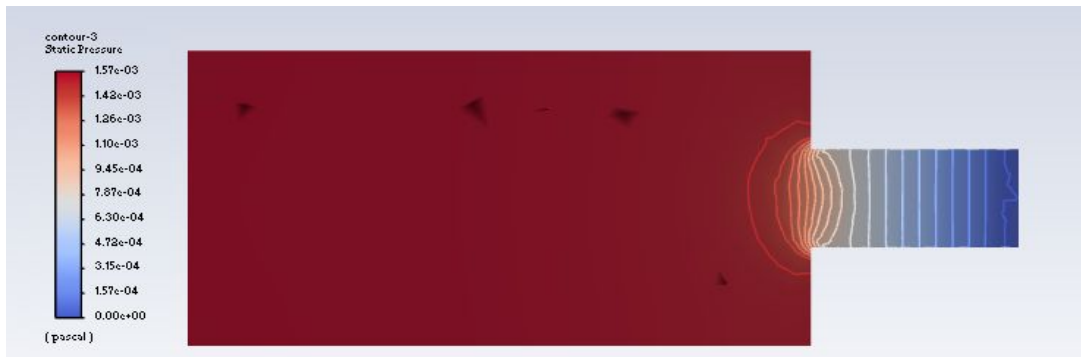


Figure 11: Line plot for V_{out} v/s flow time

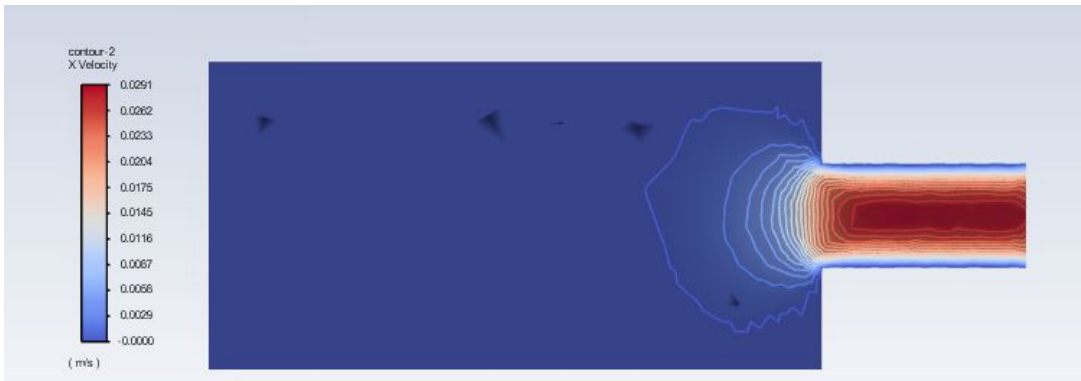
Deliverable (2)

The pressure contour, velocity magnitude contour, temperature contour and the density contour are shown at the vertical plane of the symmetry at $t = 5$ s, in Fig. 12

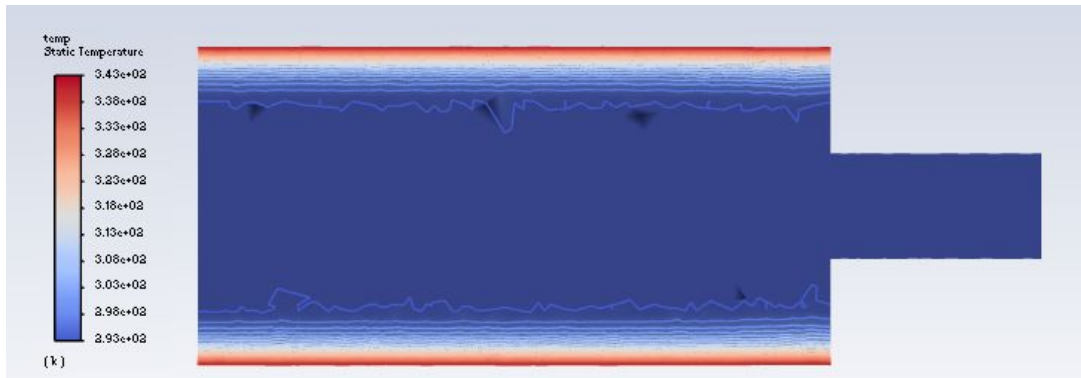
(a), (b) (c) and (d) respectively.



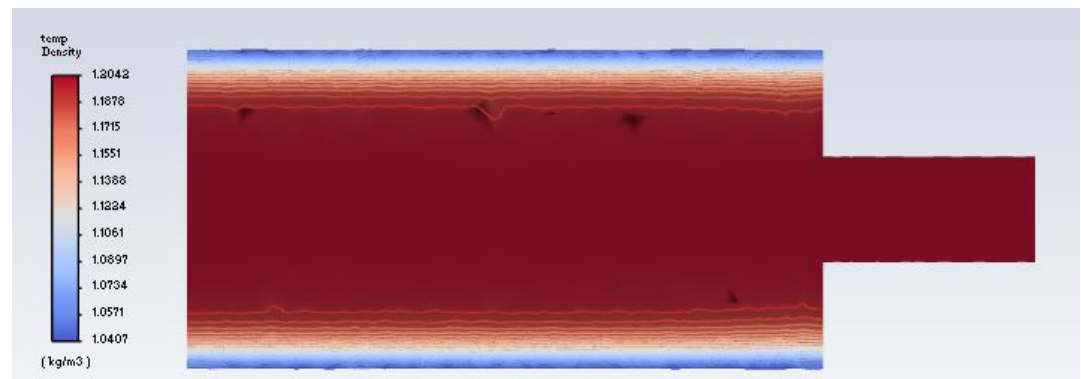
(a)



(b)



(c)



(d)

Figure 12: (a) Pressure Contour, (b) Velocity magnitude contour and (c) Temperature contour (d) density contour at the plane of symmetry.

Deliverable (3)

1. The fractional variation of density across the system is defined by the expression given below:

$$\text{Fractional Variation of density} = (Q_{max} - Q_{min}) / \bar{Q} \quad (5)$$

Where Q_{max} = maximum density, Q_{min} = minimum density, \bar{Q} = Average density.

For the system in Task 3, the maximum and minimum density values after $t = 5$ s can be expressed as

$$Q_{max} = 1.2042 \text{ kg/m}^3, Q_{min} = 1.0407 \text{ kg/m}^3, \bar{Q} = 1.18 \text{ kg/m}^3$$

The final fractional variation of density as per equation 5 is **0.1383**. (Task3).....(i)

2. **for Task1**, as per the boussinesq approximation, the density can be estimated using equation 1 and the values are:

$$Q_{max} = 998.203 \text{ kg/m}^3, Q_{min} = 985.603 \text{ kg/m}^3, \bar{Q} = 993.1449 \text{ kg/m}^3$$

The final fractional variation of density as per equation 5 is **0.01268**. (Task1)... (ii)

3. Again for Task1, using thermal expansion coefficient beta,

$$\beta = 0.00036621, \text{ Temperature difference} = 55 - 20 = 35 \text{ degree celsius}$$

Fractional variation of density = **0.0128** (close to the task1 value)

Comparing (i) and (ii), the task 3 value is higher than task 1 because task3 uses ideal gas with a volumetric expansion where density varies with temperature. On the other hand, for Task 1, the fluid considered as was water with variation of density with temperature and buoyancy effect included. For task 1, the density is not a string function of temperature. Therefore, the fractional density is lower in task1 when compared to task 3.

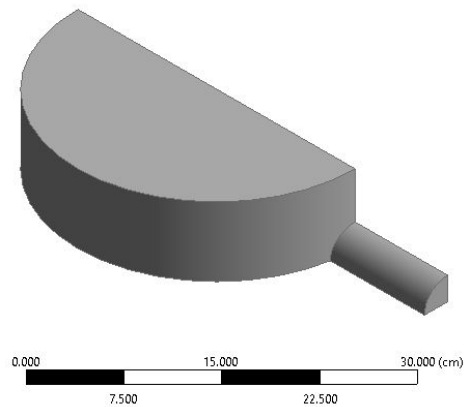
Task 4

To perform the task 3 invoking the concept of symmetry in 2 planes.

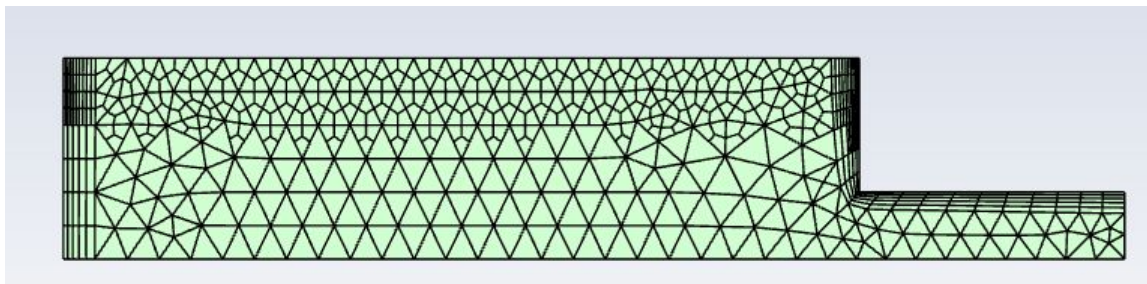
Solution

Deliverable (3)

Figure 13 (a) shows the computational domain of the quarter chamber and Fig. 13 (b) shows the local mesh refinement at the symmetry used in the present simulation.



(a)



(b)

Figure 13: (a) Computational domain for the quarter chamber, (b) meshed surface at the symmetry with local mesh refinement

Deliverable (1)

The line plot for V_{out} m/s time from $t = 0$ s to $t = 5$ s is shown in Fig. 14. As the temperature of the air increases in the chamber it expands and as a result a continuous increase in the velocity (area weighted average) of the air, normal to the outlet, is observed. The setup remains the same as the above task 3. The value of V_{out} is 0.016848 (m/s) at $t = 5$ s.

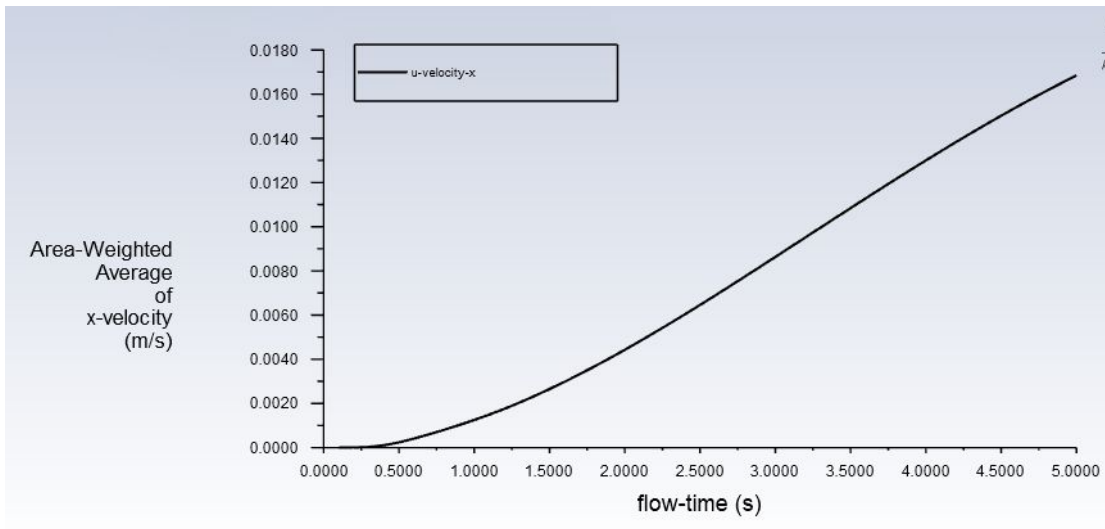
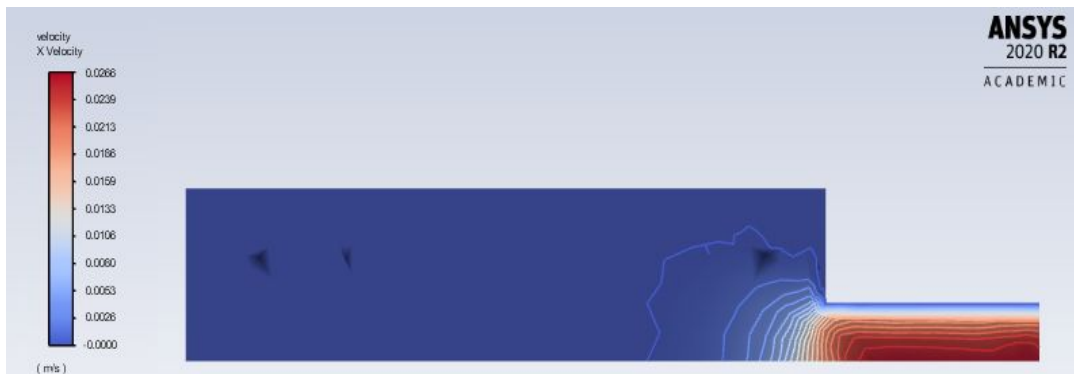


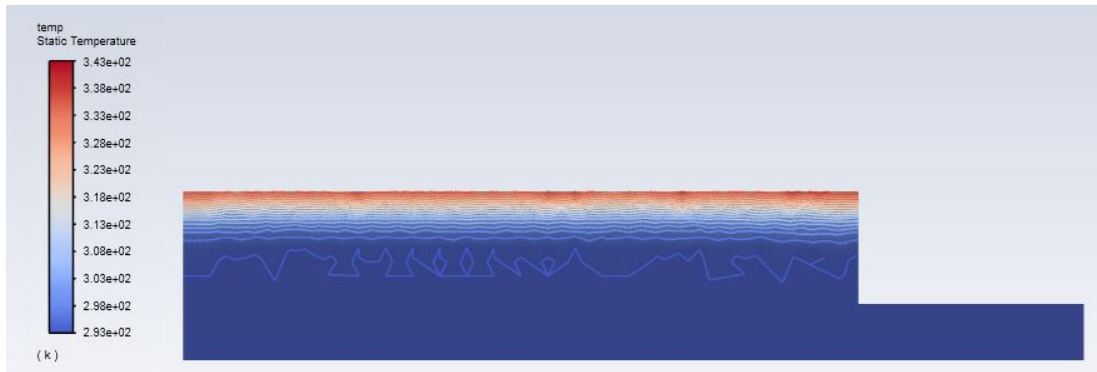
Figure 14: Line plot for V_{out} v/s flow time t from $t = 0$ s to $t = 5$ s.

Deliverable (2)

The velocity magnitude contour and the temperature contour are shown at the plane of the symmetry at $t = 5$ s, in Fig. 15 (a) and (b) respectively. It is discernible that the results obtained in the present simulation over a quarter domain do corroborate with that obtained in task 3. However, there are very minor discrepancies in the result which are caused mainly due to difference between the grid generated in original computational domain and the grid generated in the quarter domain, and in addition to that, also due to some numerical error.



(a)



(b)

Figure 15: (a) Velocity magnitude contour and (b) Temperature contour at the plane of symmetry.

Deliverable (4)

A brief discussion to compare Task 4 and Task 3. Are the results the same? What are the savings by adopting the two-fold symmetry, in terms of number of elements and computational time?

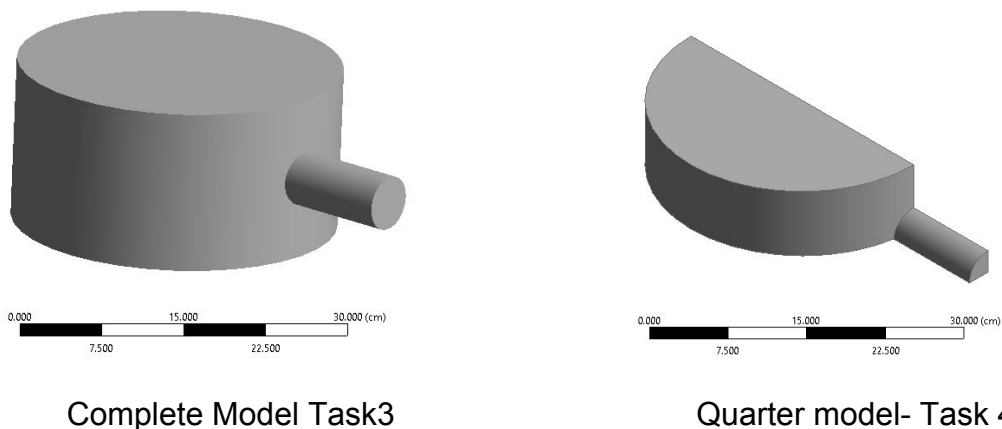


Figure 16: Complete model and two-fold symmetry model

Comparing Task 3 and Task 4 results are approximately the same with a variation of 5-10%. In Task 4, the symmetric model is used to evaluate the same results under boundary conditions. The symmetric model is a quarter of the full model results in reduction of number of elements and nodes significantly as shown below.

	Nodes	elements
Complete Model	13725	42891
Quarter Model	4217	13215

Table 2: Number of Nodes and elements for Complete and Quarter model

It can be seen through Table 2 that for the quarter model the nodes and elements are

almost reduced to one fourth of the initial model. The computational equations are solved at the elements and nodes to produce the required results. It can be concluded that higher the number of elements and nodes, the more will be the number of equations points to be solved in CFD. Therefore, the quarter model takes about one fourth of the computational time taken by complete model as shown below:

Performance Timer for 2500 iterations on 2 compute nodes		Performance Timer for 2500 iterations on 2 compute nodes	
Average wall-clock time per iteration:	0.050 sec	Average wall-clock time per iteration:	0.162 sec
Global reductions per iteration:	91 ops	Global reductions per iteration:	92 ops
Global reductions time per iteration:	0.000 sec (0.0%)	Global reductions time per iteration:	0.000 sec (0.0%)
Message count per iteration:	163 messages	Message count per iteration:	187 messages
Data transfer per iteration:	0.269 MB	Data transfer per iteration:	0.871 MB
LE solves per iteration:	1 solves	LE solves per iteration:	1 solves
LE wall-clock time per iteration:	0.024 sec (48.5%)	LE wall-clock time per iteration:	0.081 sec (50.2%)
LE global solves per iteration:	1 solves	LE global solves per iteration:	1 solves
LE global wall-clock time per iteration:	0.000 sec (0.1%)	LE global wall-clock time per iteration:	0.000 sec (0.0%)
LE global matrix maximum size:	24	LE global matrix maximum size:	18
AMG cycles per iteration:	1.000 cycles	AMG cycles per iteration:	1.001 cycles
Relaxation sweeps per iteration:	52 sweeps	Relaxation sweeps per iteration:	64 sweeps
Relaxation exchanges per iteration:	0 exchanges	Relaxation exchanges per iteration:	0 exchanges
LE early protections (stall) per iteration:	0.000 times	LE early protections (stall) per iteration:	0.000 times
LE early protections (divergence) per iteration:	0.000 times	LE early protections (divergence) per iteration:	0.000 times
Total SVARS touched:	370	Total SVARS touched:	370
Time-step updates per iteration:	0.02 updates	Time-step updates per iteration:	0.02 updates
Time-step wall-clock time per iteration:	0.000 sec (0.2%)	Time-step wall-clock time per iteration:	0.000 sec (0.1%)
Total wall-clock time:	125.289 sec	Total wall-clock time:	405.562 sec

125.289 sec

405.562 sec

Quarter Model Task3

Complete model- Task 4

Figure 17: Computational time for both models

The time saved is almost 280 s i.e 69% of the total time taken by the complete model. Hence, the symmetry saves a significant time in solving the huge models and requires less expenditure in industries where minute percentage errors are acceptable.