MAE 560 - Applied Computational Fluid Dynamics

Project 2

Sean Haynie

November 12, 2021

Dr. H.P. Huang

Name of Collaborator: *Danielle Seddon*	
Task(s), specific detail	Contribution or collaborative effort
Task 4	*Worked on MATLAB code together*

Task 1

Task 1a: Three boundaries set to pressure outlet

This task simulates the leaking of natural gas from an underground vault into open air, in a pure 2-D setting. This simulation has the lateral and top boundaries set to pressure outlets with zero gauge pressure and open to air. A transient solution to t = 5 s was performed and the results are included below.

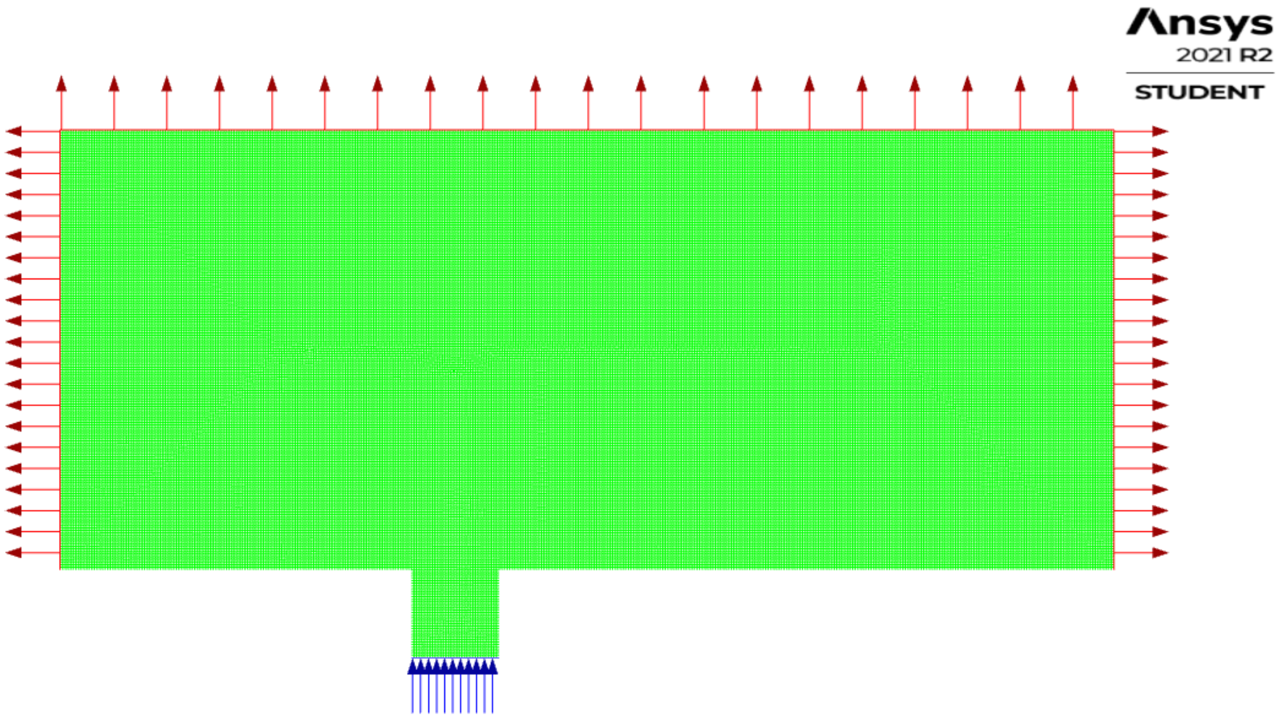

Figure 1-1: Plot of the mesh used in Task 1a

Deliverable (D1)

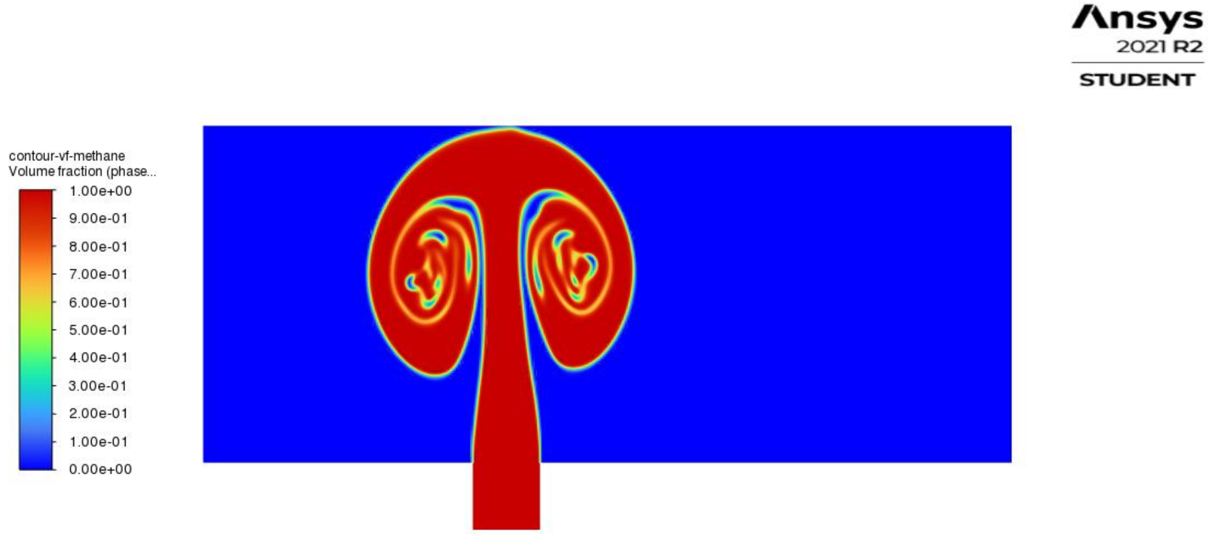

Figure 1-2: Contour plot of the volume fraction of methane at t = 5 s

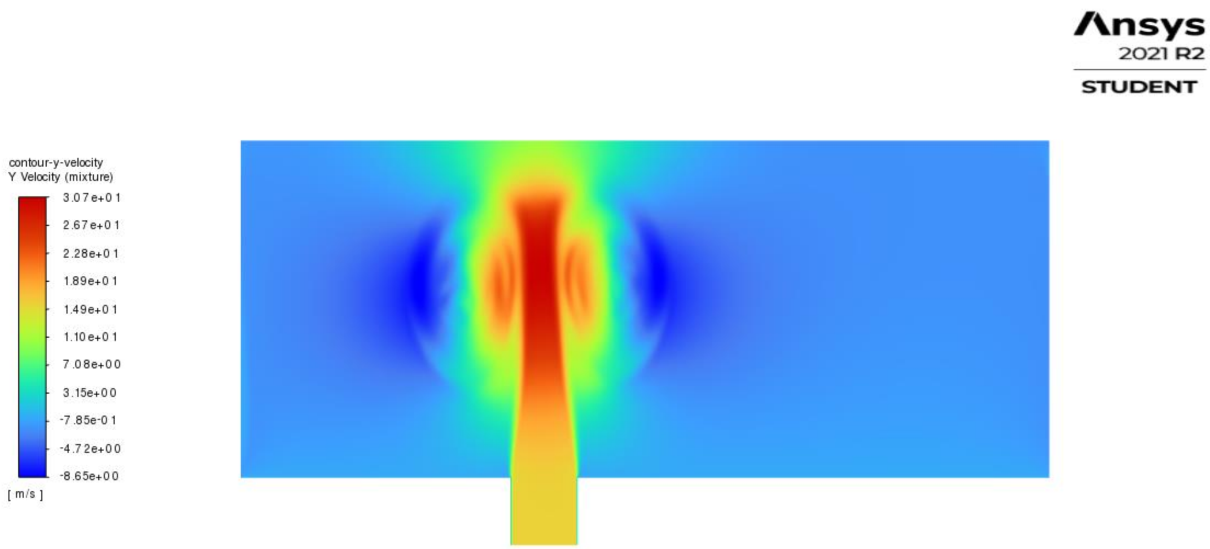

Figure 1-3: Contour plot of the y-velocity at t = 5 s

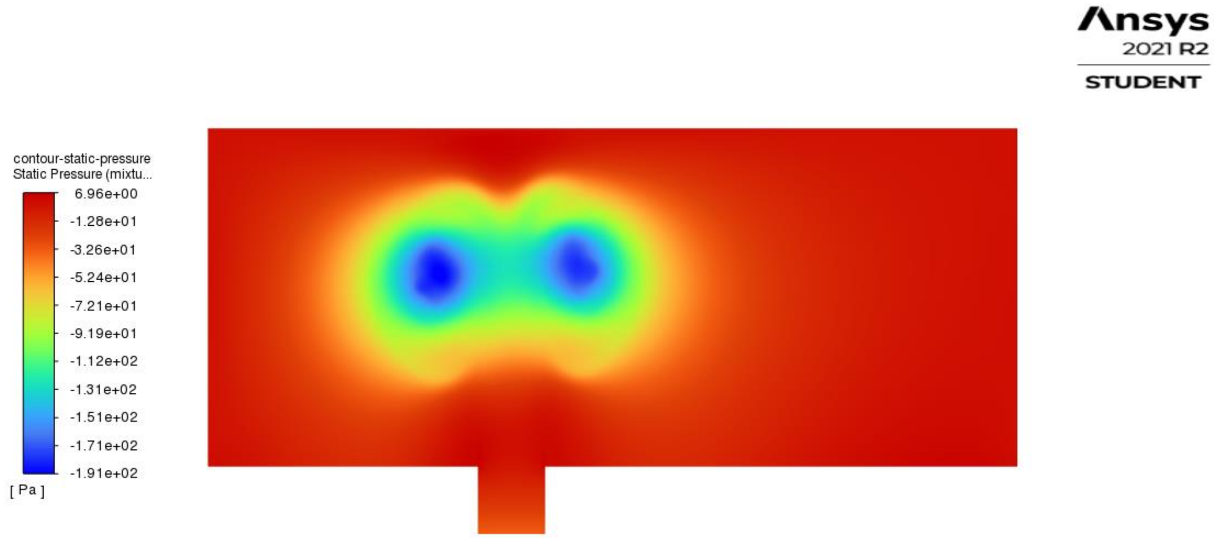

Figure 1-4: Contour plot of the static pressure at t = 5 s

Task 1b: Left boundary set to velocity inlet

This task uses the same settings as the previous one, however the left boundary wall is set to a velocity inlet with the x-velocity profile given by $u = 0.8\ y - 0.032\ y^2$. The transient solution was performed to t = 5 s and the results are included below.

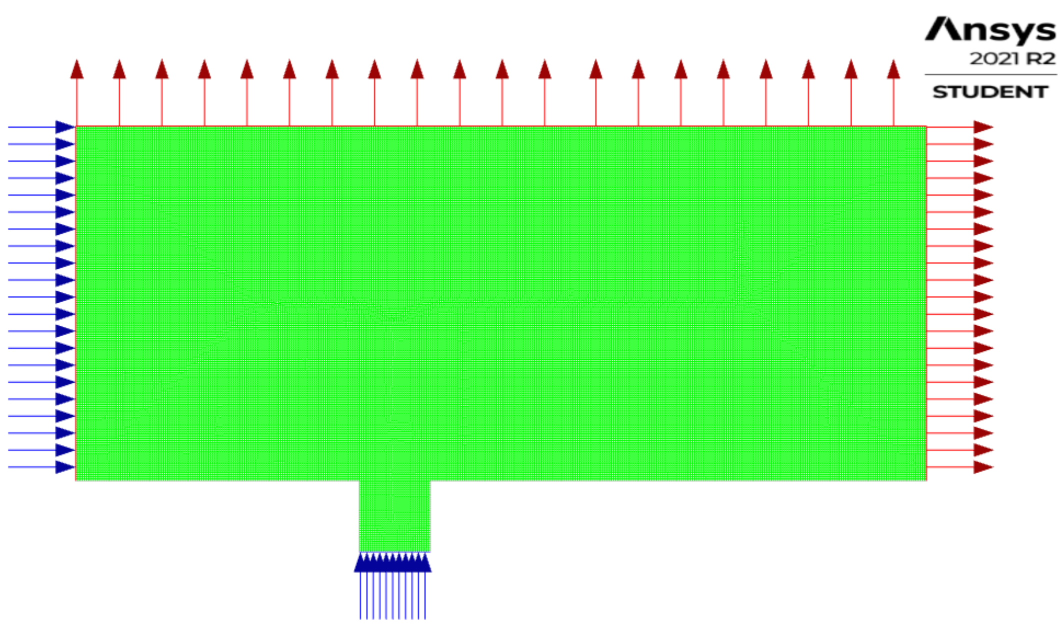

Figure 1-5: Plot of the mesh used in Task 1b

Deliverable (D2)

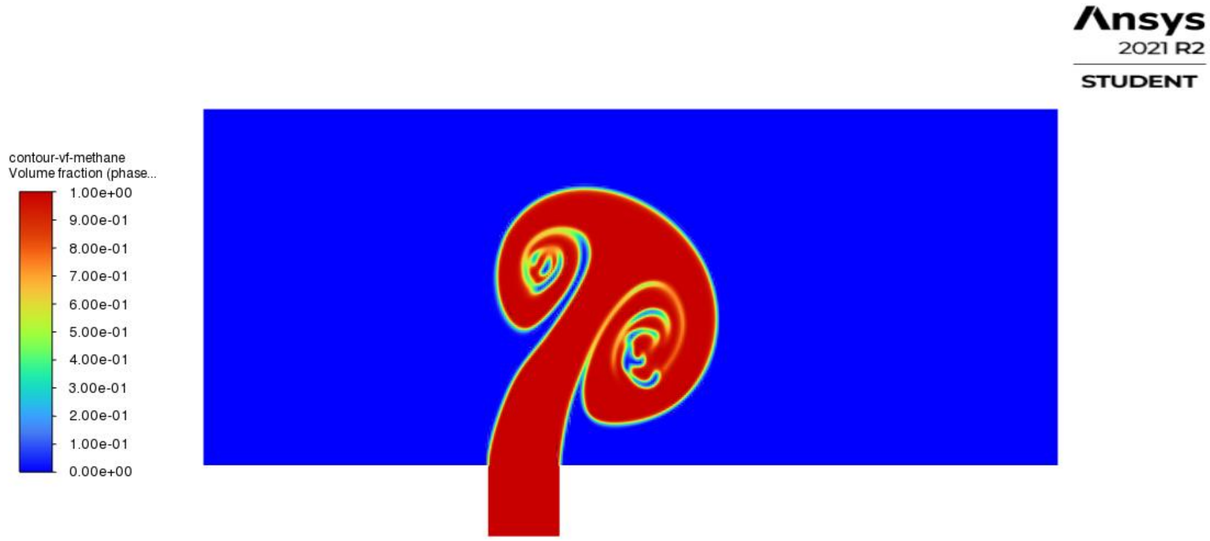

Figure 1-6: Contour plot of the volume fraction of methane at t = 5 s

Deliverable (D3)

The mesh resolution parameters and time step size used for both Tasks 1a and 1b are included in the table below.

Table 1: Mesh resolution parameters and time step setup for Tasks 1a and 1b

Element Size (m)	**Number of Elements**	**Time Step Size (s)**	**Number of Time Steps**	**Max Iterations/Time Step**
0.15	67,813	0.01	500	10

Task 2

This task simulates the process of a falling water droplet impacting on a flat water surface, in a pure 2-D setting. A 50 cm x 50 cm square bucket is open to air at the top with zero gauge pressure, with the other three sides being walls, and is filled with water up to 15 cm. A circular droplet with a diameter of 6 cm is placed in the middle of the bucket, with its center placed 40 cm above the floor, and will begin to fall at t > 0. A laminar model is used, and the transient solution is performed to t = 0.3 s. The results of this simulation are included below.

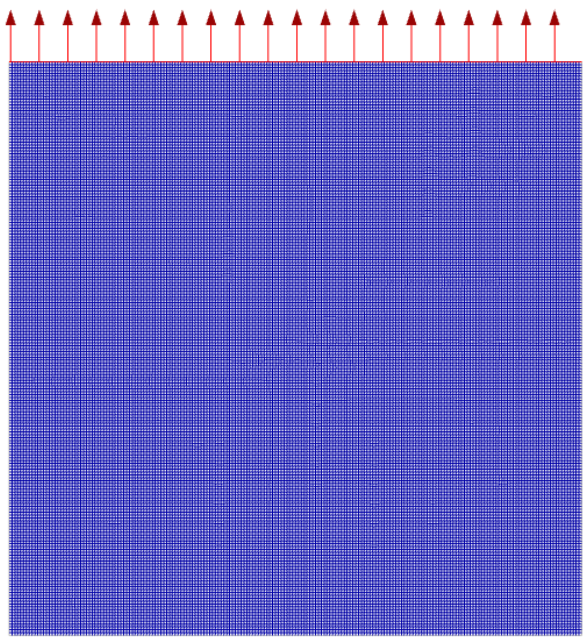

Figure 2-1: Plot of the mesh used in Task 2

Deliverable (D5)

The mesh resolution parameters and time step size used for Task 2 are included in the table below.

Table 2: Mesh resolution parameters and time step setup for Task 2

Element Size (cm)	**Number of Elements**	**Time Step Size (s)**	**Number of Time Steps**	**Max Iterations/Time Step**
0.25	40,000	0.001	300	10

Deliverable (D4)

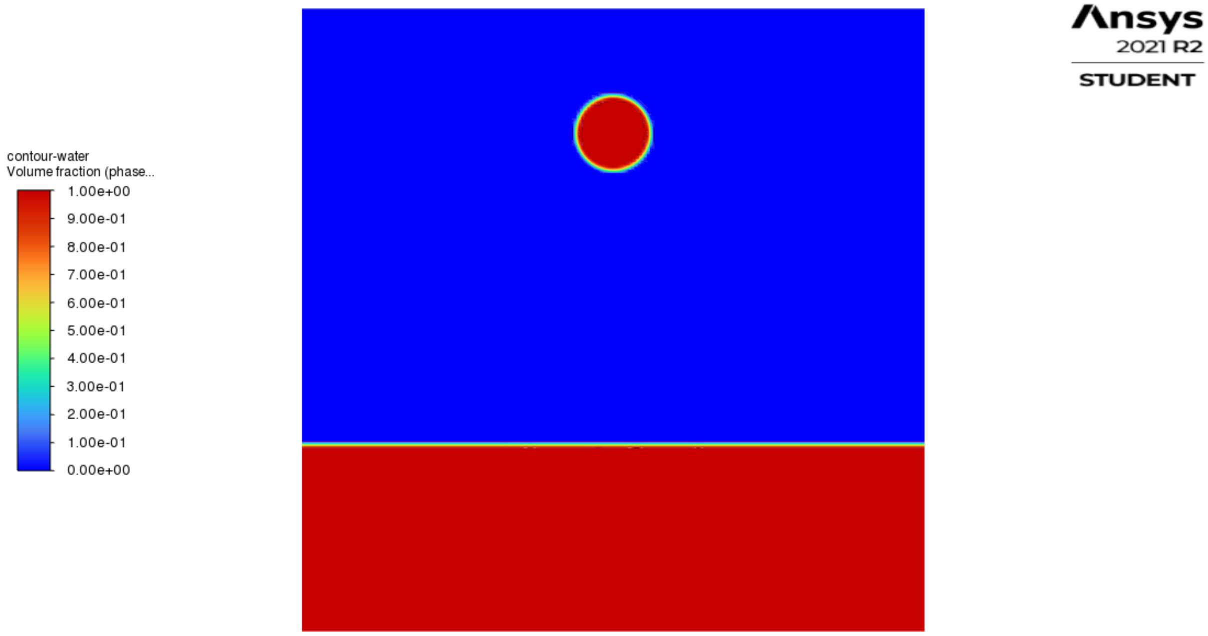

Figure 2-2: Contour plot of the volume fraction of water at t = 0 s

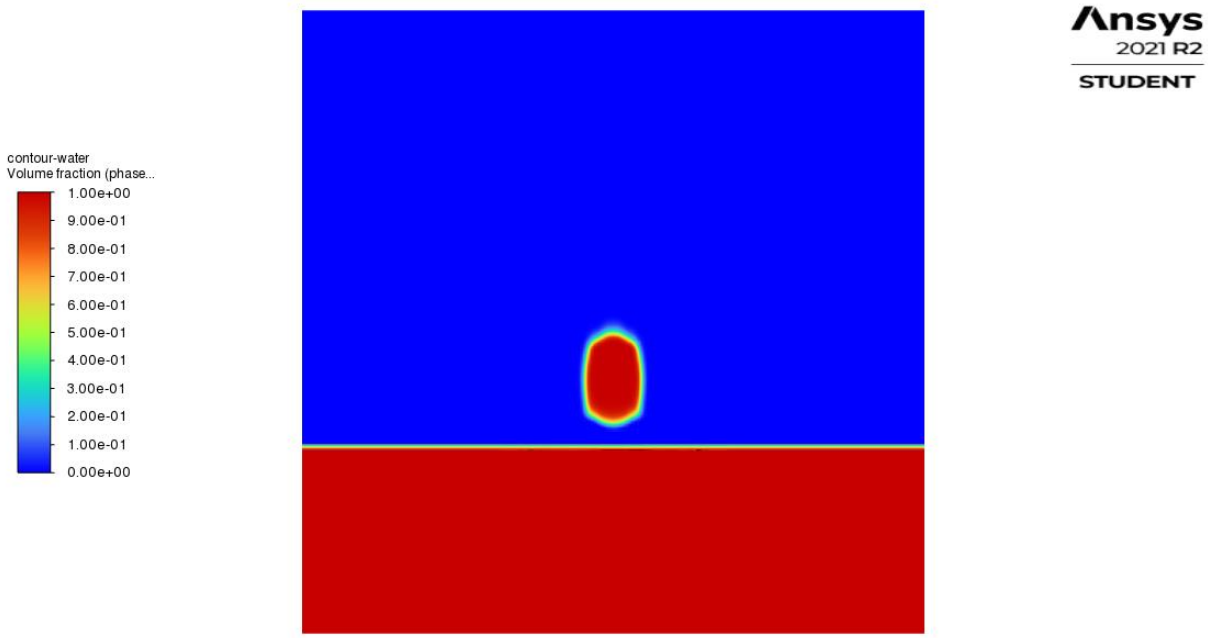

Figure 2-3: Contour plot of the volume fraction of water at t = 0.2 s

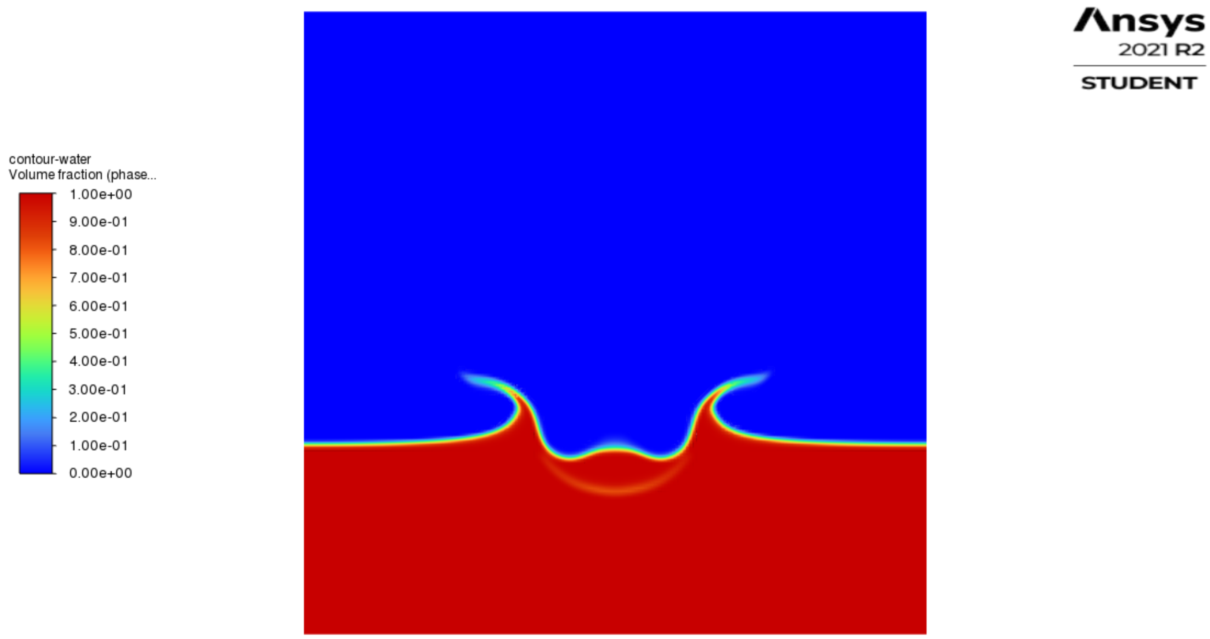

Figure 2-4: Contour plot of the volume fraction of water at t = 0.25 s

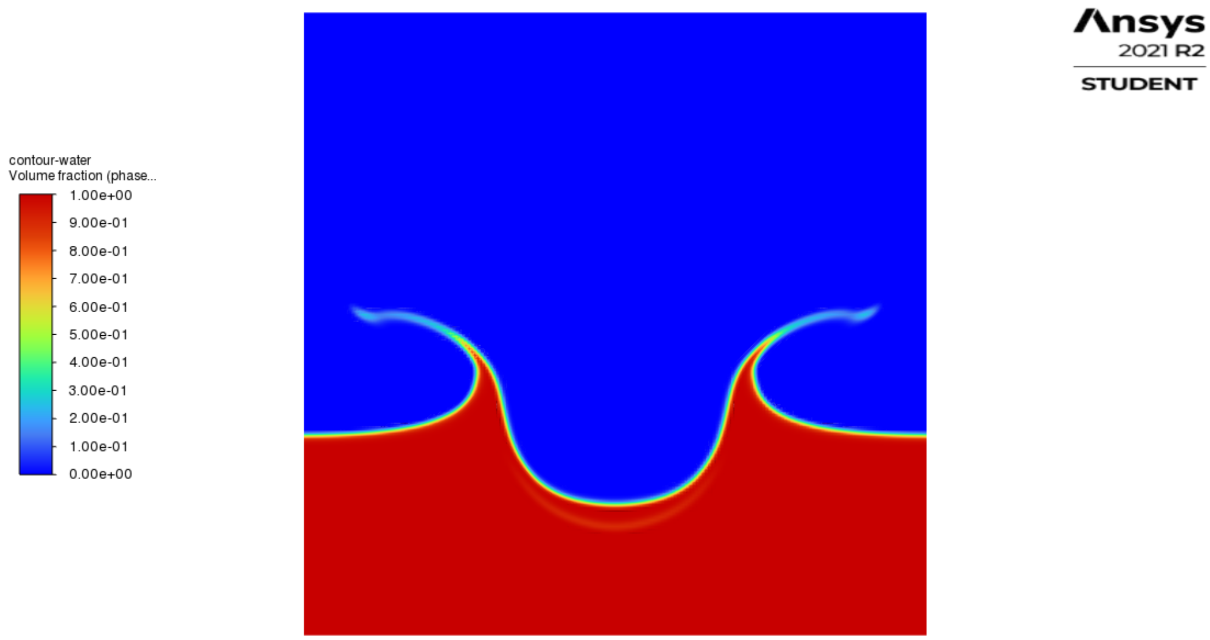

Figure 2-5: Contour plot of the volume fraction of water at t = 0.3 s

Task 3

This task simulates a droplet of engine oil sliding down a 45 degree incline surface. A rectangular domain was created, with the bottom plate being set to a wall and the others being open to air. The details and results of this simulation are shown below.

Deliverable (D6)

The computational domain for this simulation is a 6 cm x 12 cm x 2 cm rectangle. The bottom plate was set to a wall, and the other boundary conditions were set to a pressure outlet with zero gauge pressure and backflow set to air. The mesh resolution parameters and time step size used for Task 3 are included in the table below.

Table 3: Mesh resolution parameters and time step setup for Task 3

Element Size (cm)	Number of Elements	Time Step Size (s)	Number of Time Steps	Max Iterations/Time Step
0.10	120,000	0.0002	500	10

Deliverable (D7)

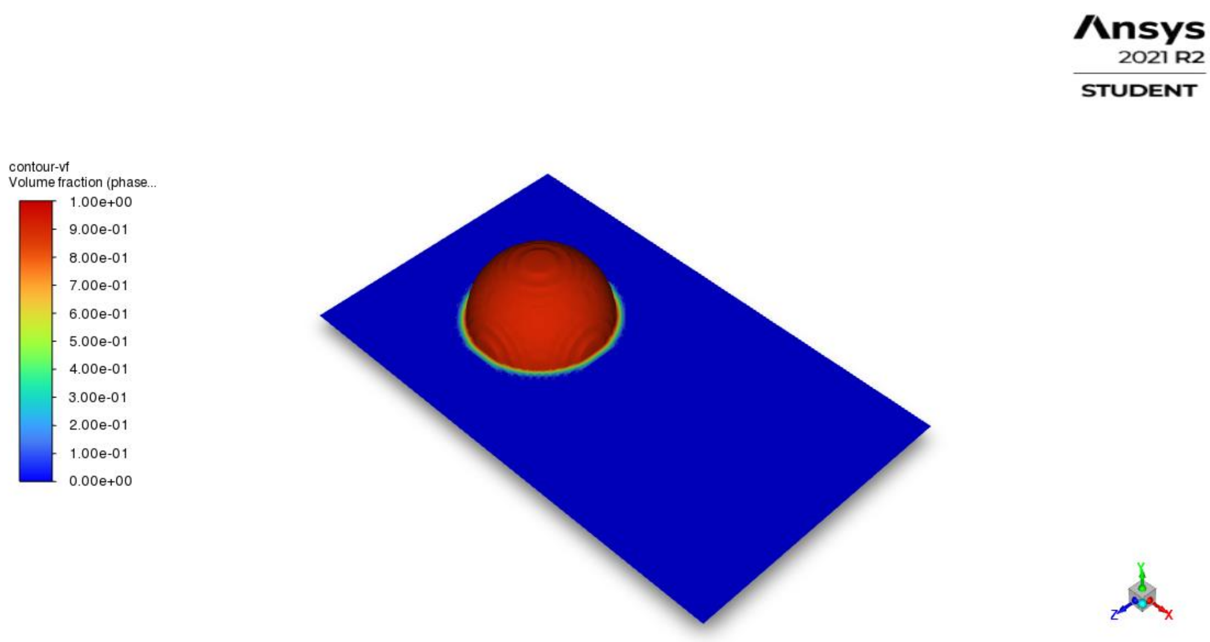

Figure 3-1: 3-D shape of the blob of engine oil at t = 0 s

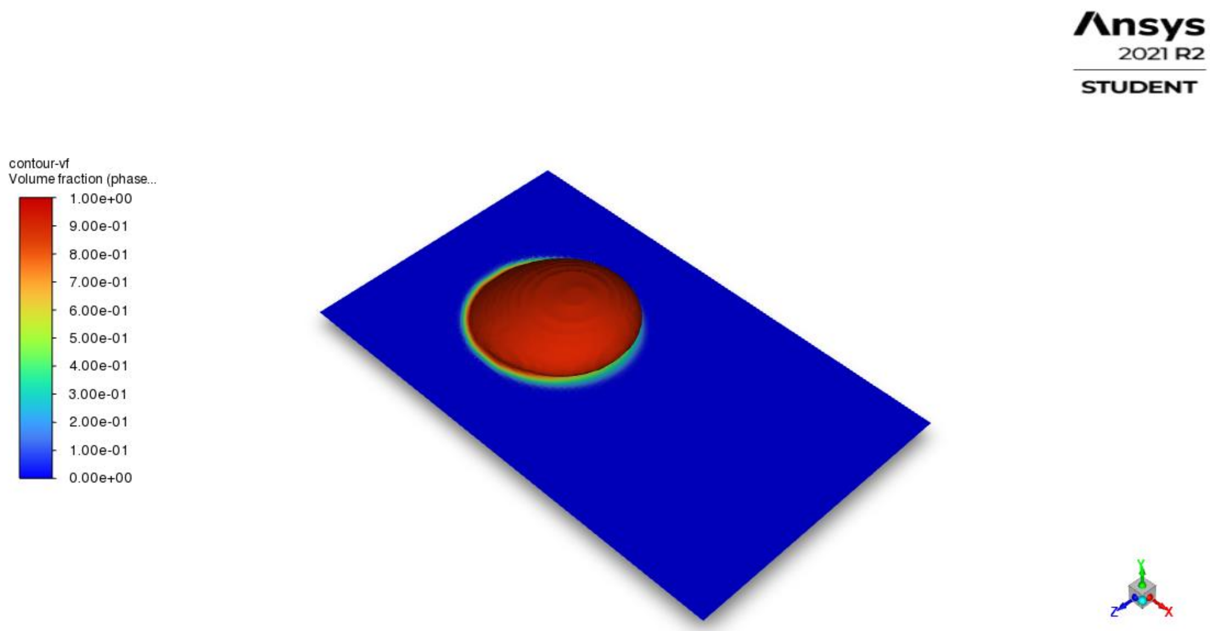

Figure 3-2: 3-D shape of the blob of engine oil at t = 0.05 s

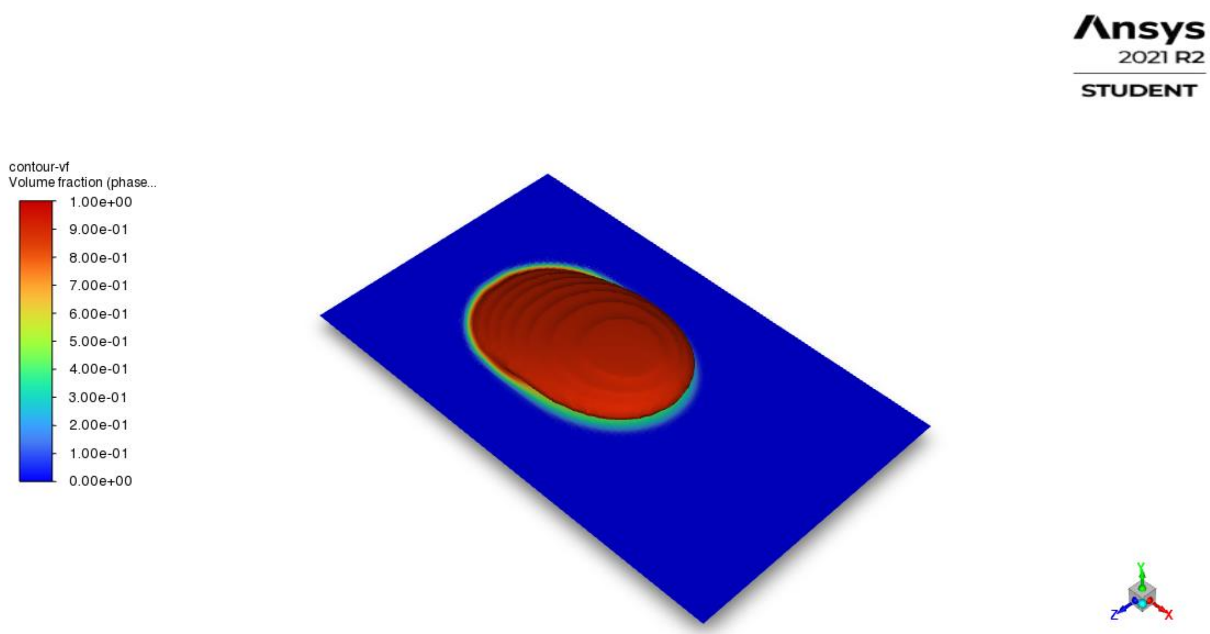

Figure 3-3: 3-D shape of the blob of engine oil at t = 0.1 s

Deliverable (D8)

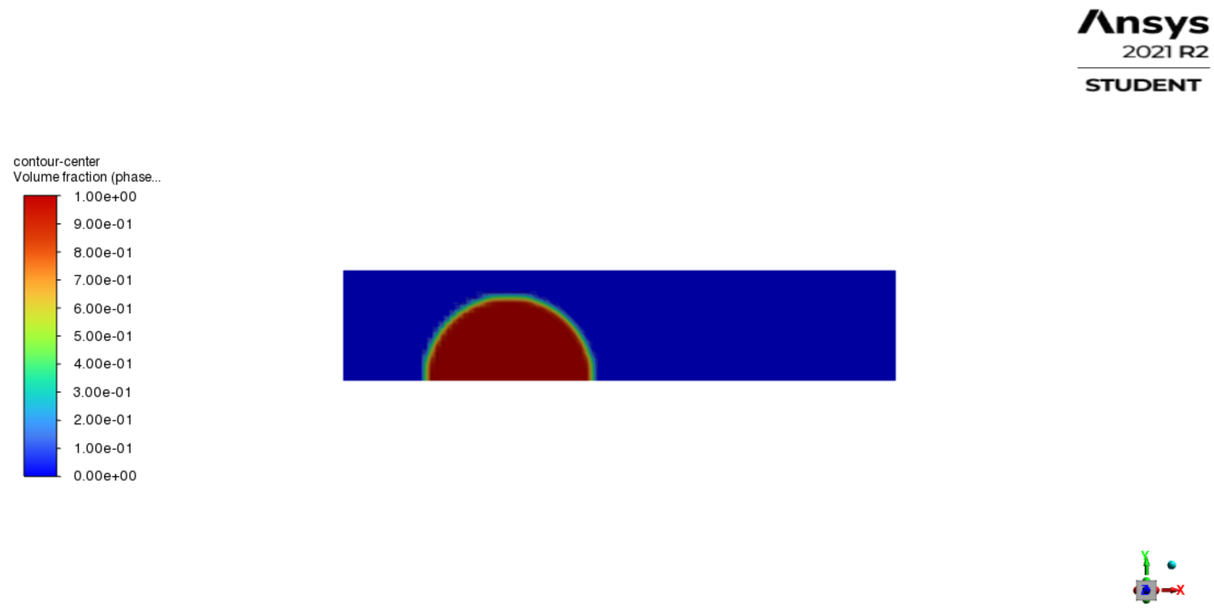

Figure 3-4: Contour plot of the volume fraction of engine oil at t = 0 s

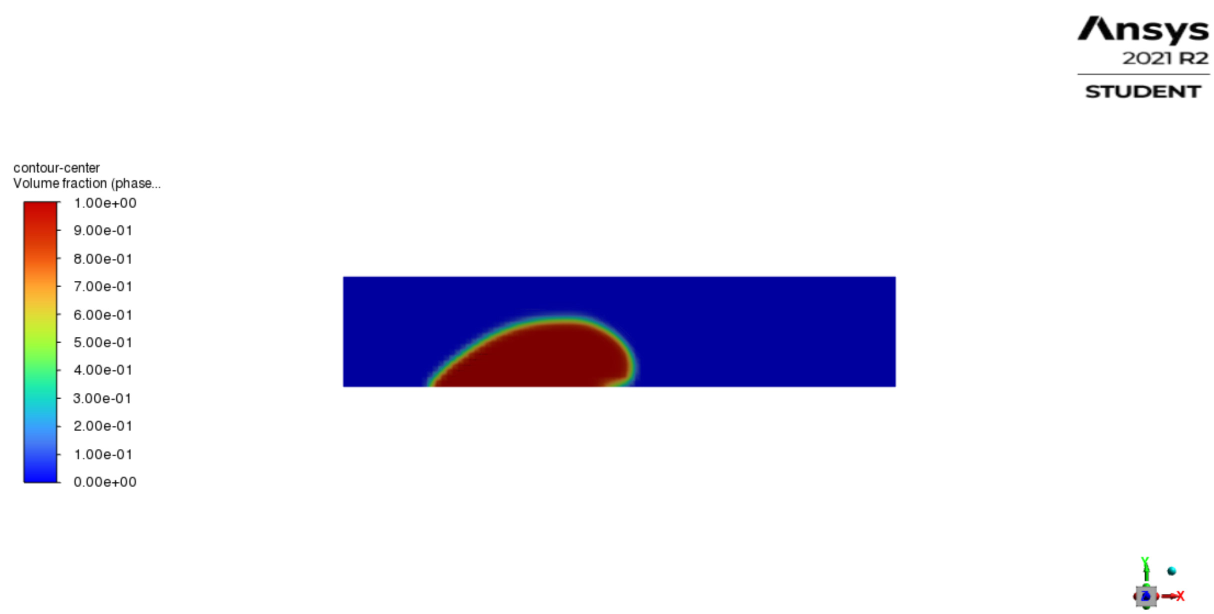

Figure 3-5: Contour plot of the volume fraction of engine oil at t = 0.05 s

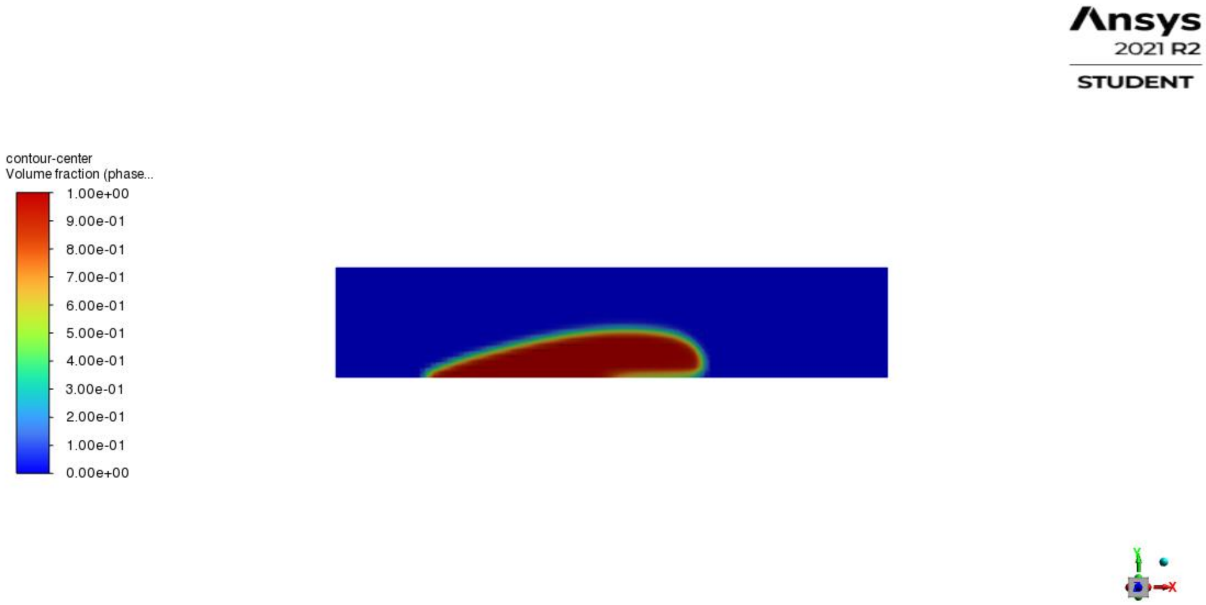

Figure 3-6: Contour plot of the volume fraction of engine oil at t = 0.1 s

Task 4

This task simulates the flow in a U-shaped 3-D pipe with a circular cross section. Water is filled in the pipe so that the left side is 5 cm from the top opening and the right side is 15 cm from its respective opening in the top. Air is filled in the pipe above these openings. A laminar model is used, and the transient solution is performed to t = 1.25 s. The simulation was performed using a half-pipe geometry by invoking symmetry and the results are included below.

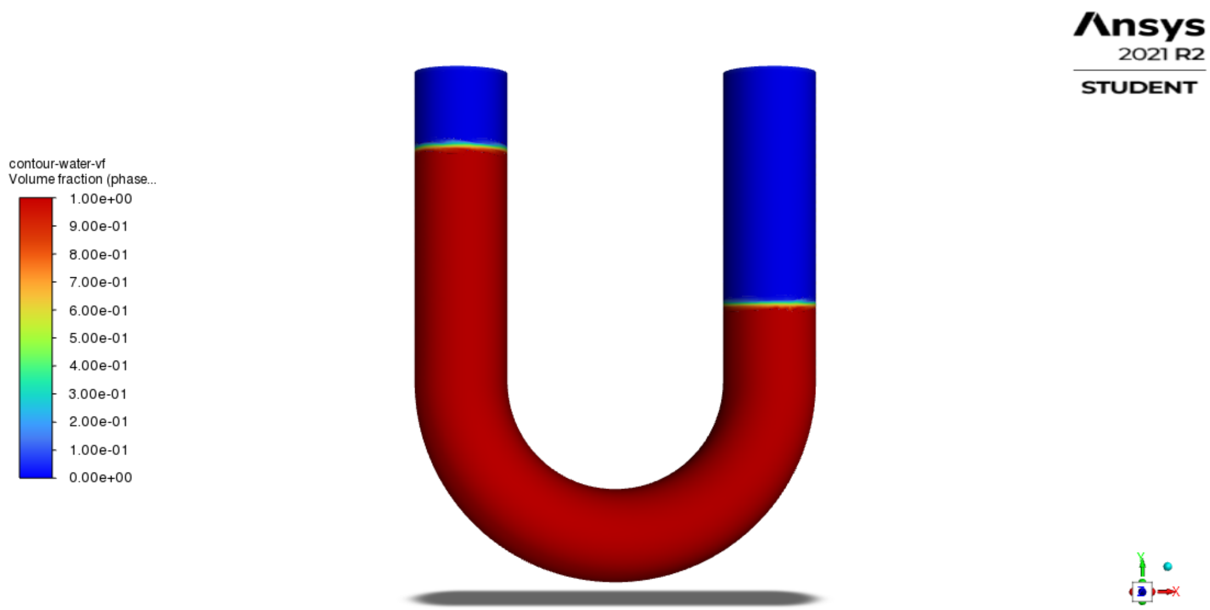

Figure 4-1: Contour plot of the volume fraction of water at t = 0 s

Deliverable (D9)

The top two openings were set to pressure outlets with backflow volume fraction of water set to 0 to allow Fluent to properly simulate the oscillation.

Table 4: Mesh resolution parameters and time step setup for Task 4

Element Size (cm)	**Number of Elements**	**Time Step Size (s)**	**Number of Time Steps**	**Max Iterations/Time Step**
0.35	84,732	0.005	250	10

Deliverable (D10)

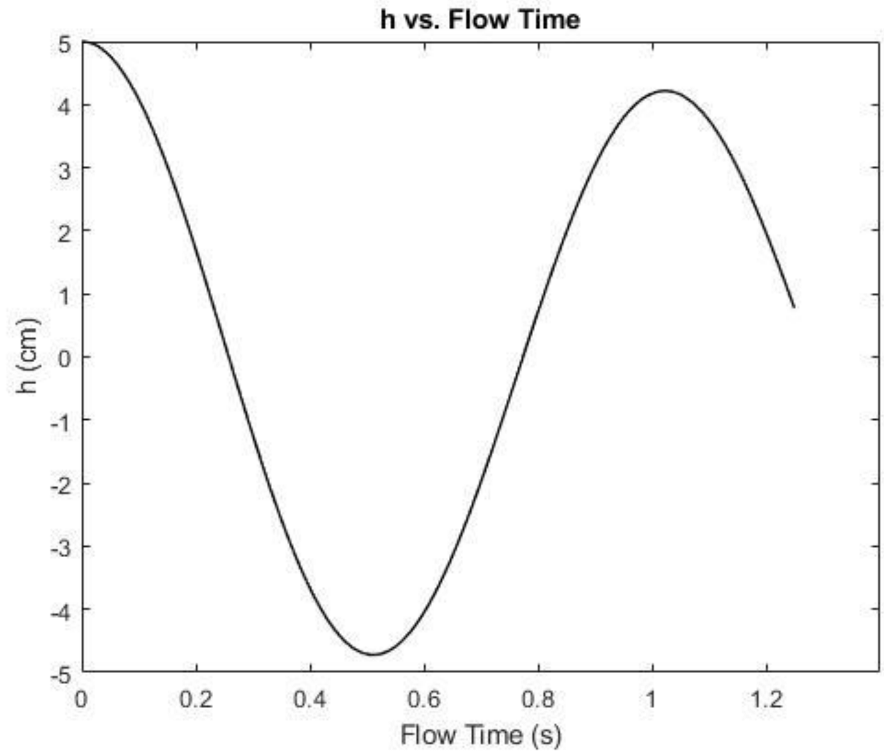

Figure 4-2: Plot of h (the water level in the left leg relative to the equilibrium level) as a function of time

From the figure above, the period of oscillation for this simulation is around 1.028 s, and therefore the time when the water levels of the left and right legs first become equal is around 0.257 s.

Deliverable (D11)

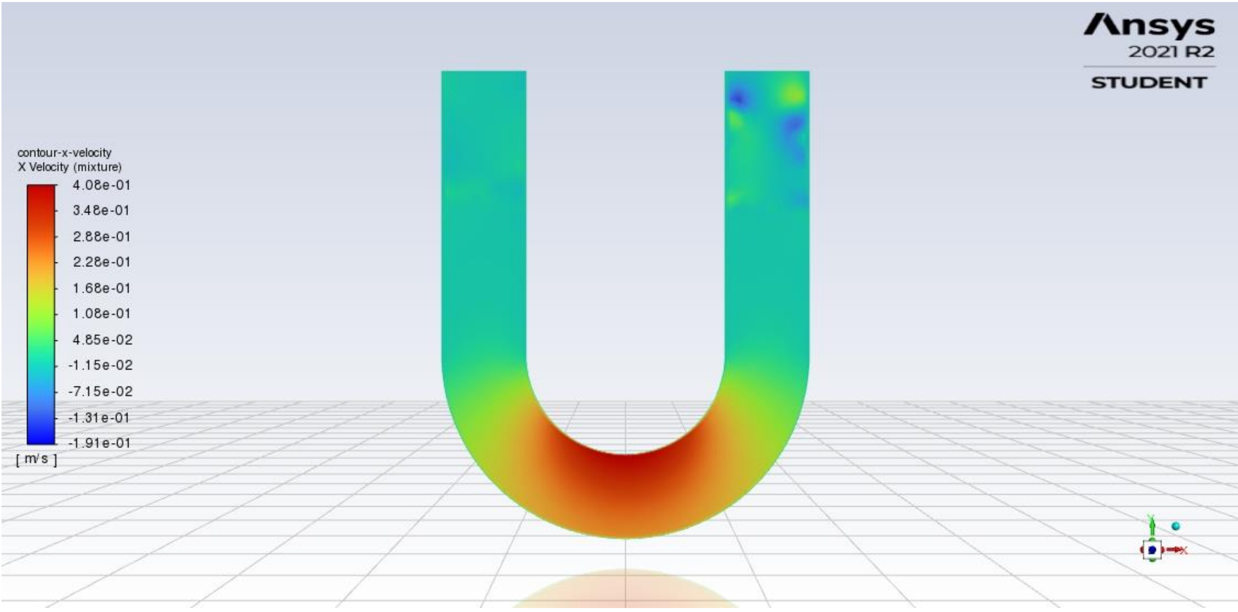

Figure 4-3: Contour plot of the x-velocity on the plane of symmetry at t = 0.257 s

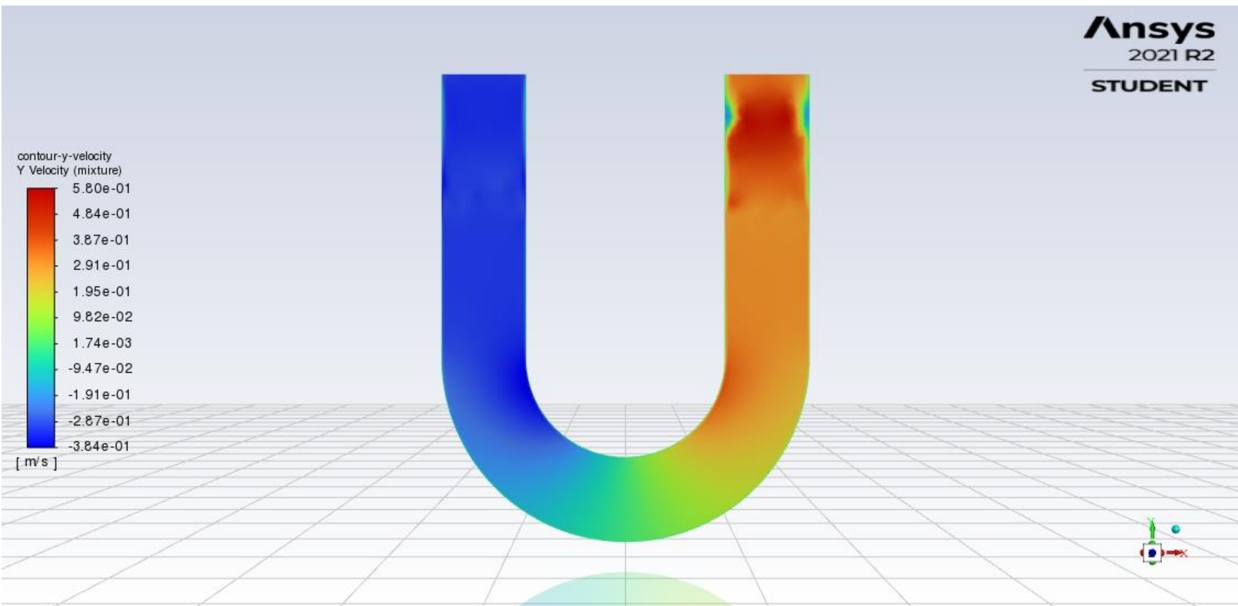

Figure 4-4: Contour plot of the y-velocity on the plane of symmetry at t = 0.257 s

From the contour plots shown above, it is evident that the maximum x-velocity is occurring at the innermost edge of the pipe, and the water is currently flowing from the left side of the pipe to the right side.