

MAE 560 Applied Computational Fluid Dynamics

## Project 2 – Multiphase Flow

Pradnesh Laxman Mhatre

ASU ID – 1226479399

Statement of Collaboration:

I declare that I had “No collaboration” with any student of the class of MAE 560: Applied Computational Fluid Dynamics for solving Project 2.

## Task 1

### Case A: Inlet of left pipe closed

(D1) Contour plots of the Volume fraction of methane at  $t = 1\text{ s}$  and  $t = 2.5\text{ s}$

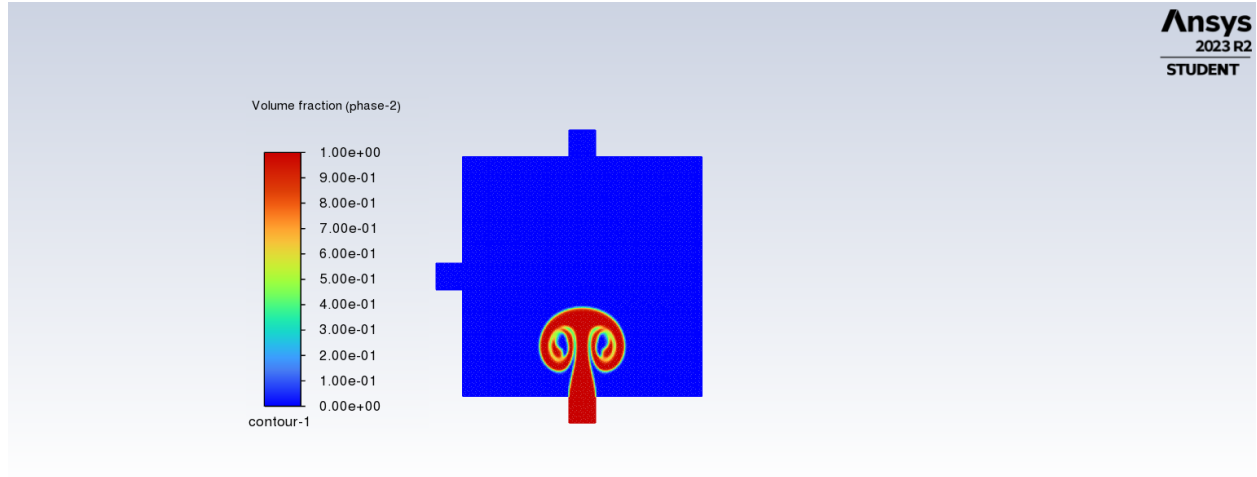


Figure 1: Contour plot of the volume fraction of methane at  $t=1\text{ s}$

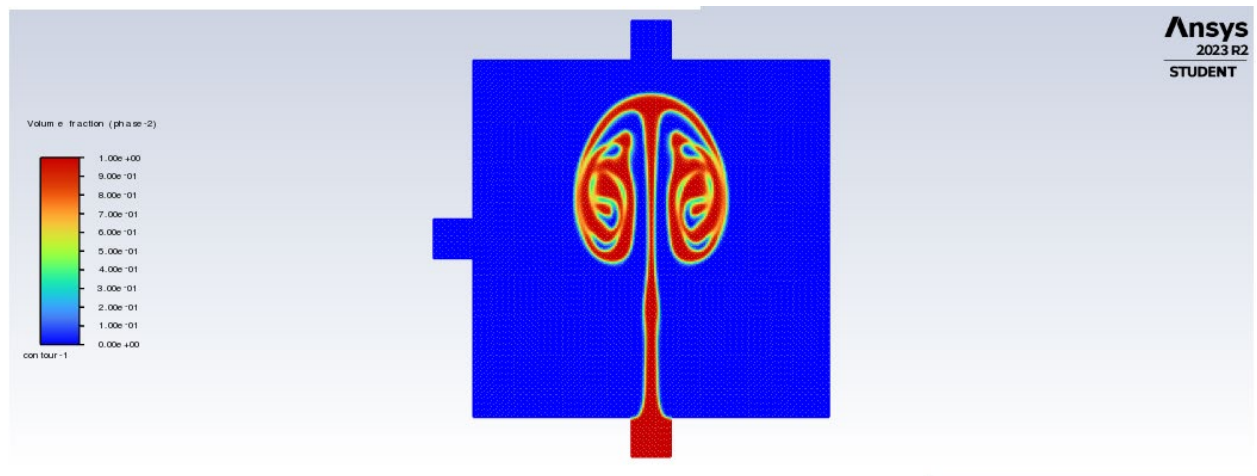


Figure 2: Contour plot of the volume fraction of methane at  $t=2.5\text{ s}$

(D2) A plot of the D-index as a function of time, for  $0 \leq t \leq 7$  s

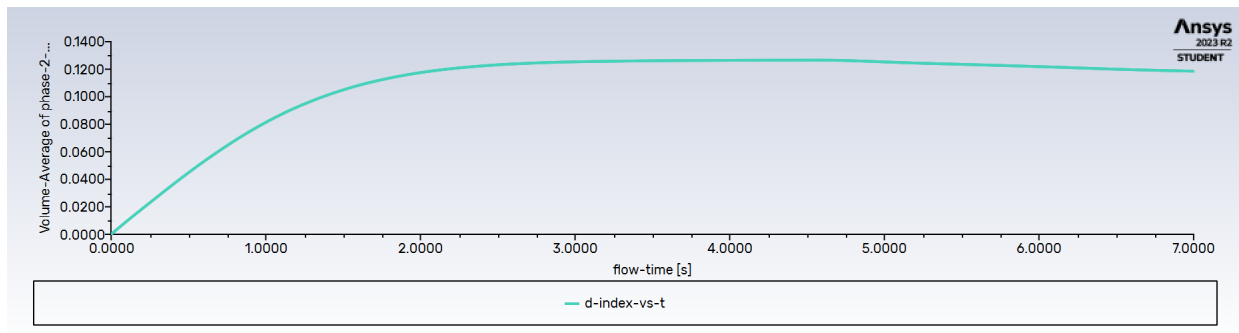
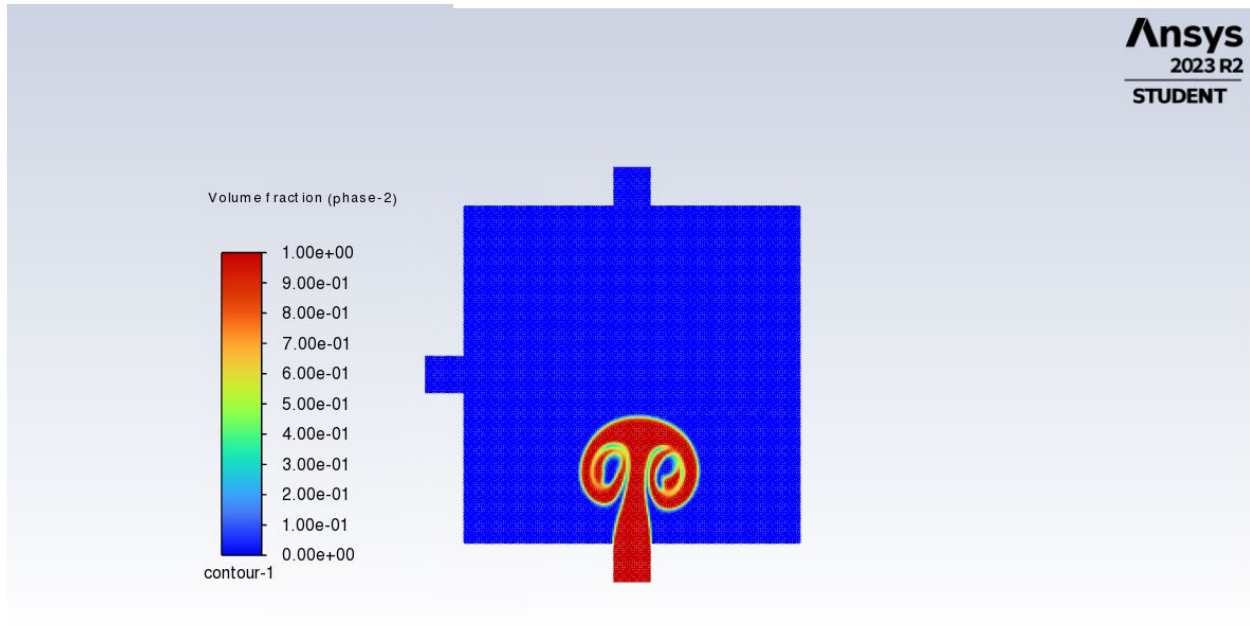


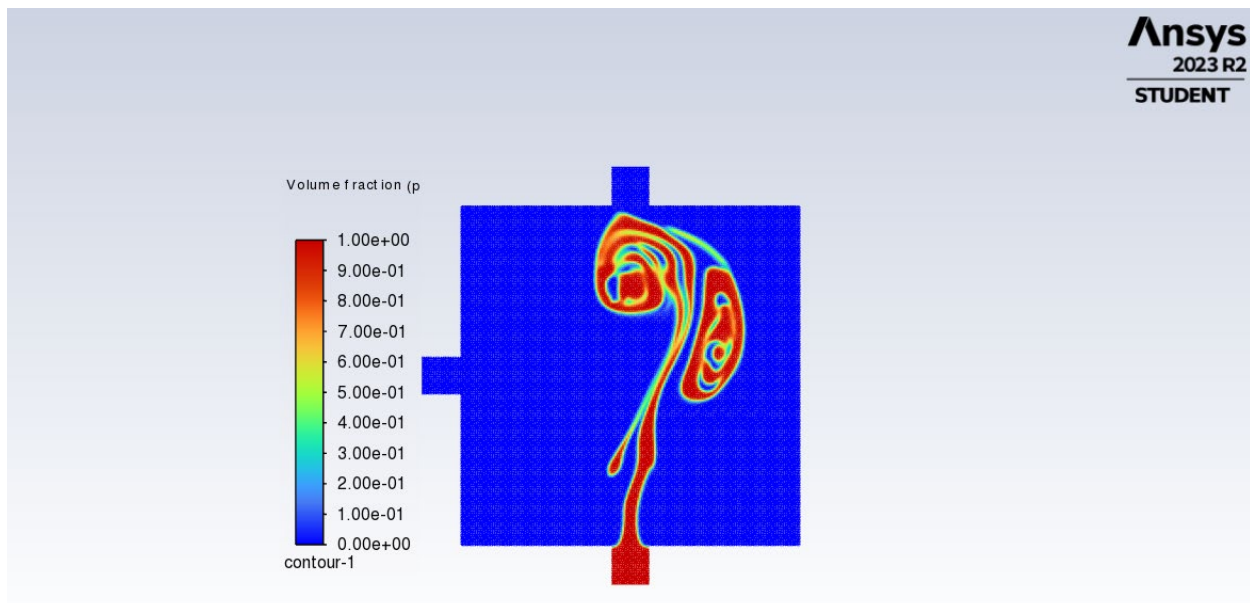
Figure 3: The D-index as a function of time, for  $0 \leq t \leq 7$  s

## Case B: Inlet of left pipe provides fresh air

(D3) Contour plots of the volume fraction of methane at  $t = 1$  s and  $t = 2.5$  s.



*Figure 4: Contour plots of the volume fraction of methane at  $t = 1$  s*



*Figure 5: Contour plots of the volume fraction of methane at  $t = 2.5$  s*

(D4) A plot of the D-index as a function of time, for  $0 \leq t \leq 7$  s. Compared to the result in (D2), does the injection of fresh air help reducing the D-index?

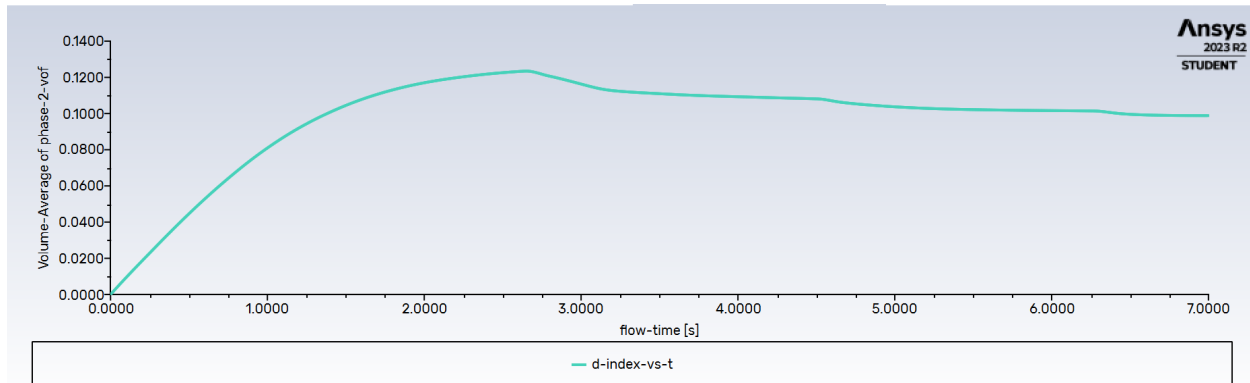


Figure 6: The D-index as a function of time, for  $0 \leq t \leq 7$  s

When we compare the D-index plot of D2 with D4 we can see a dip in graph which indicates injecting fresh air helps in reducing the D-index.

(D5) A contour plot of the velocity magnitude of the mixture at  $t = 7$  s.

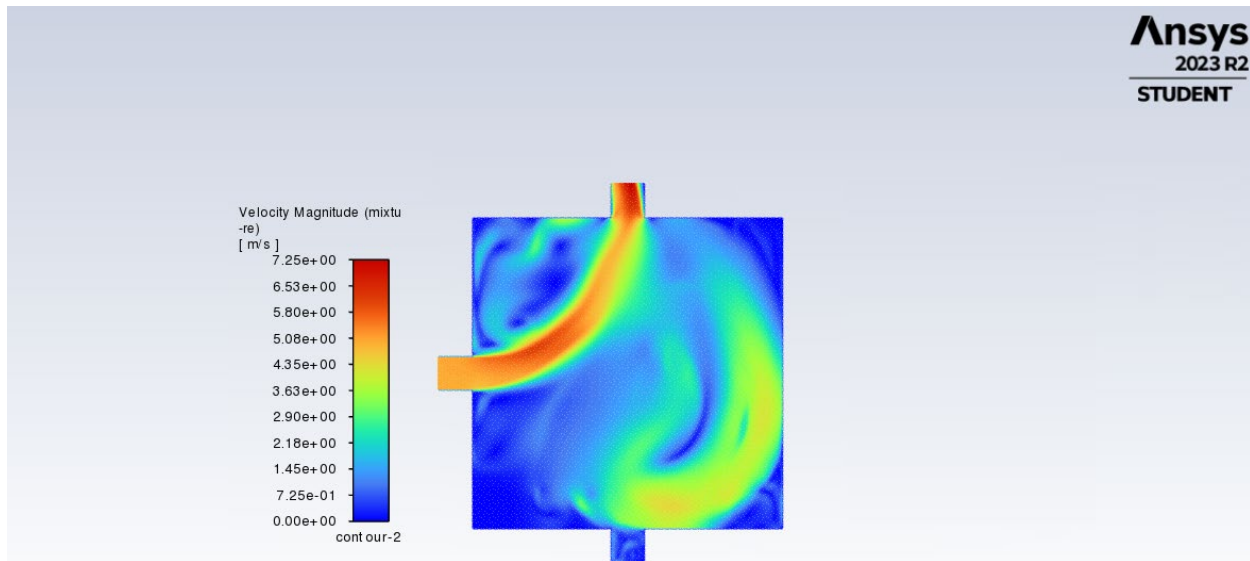


Figure 7: A contour plot of the velocity magnitude of the mixture at  $t = 7$  s

(D6) Mesh resolution and time step size used in the two simulations. If different settings are used for Case A and B, please clearly state the differences.

Mesh Resolution: 0.005 m

No. of Time Step Size: 700

Time iteration: 20

Time Step Size: 0.01

Settings were same for Case A and Case B.

## Task 2

### a) Use engine oil as the liquid

(D7) Three plots in the fashion of Fig. 3b that show the 3-D shape of the blob of engine oil at  $t = 0$ ,  $0.05$  s, and  $0.1$  s. It is part of your job to find a way to present the 3-D structure of the blob. A suggestion is to show the iso-surface of  $VF = 0.95$  where  $VF$  is the volume fraction of engine oil. This is how Fig. 3b was made. The plot should also show the bottom plate, in the fashion of Fig. 3b. This will help demonstrate how far the blob has advanced down the slope at different times.

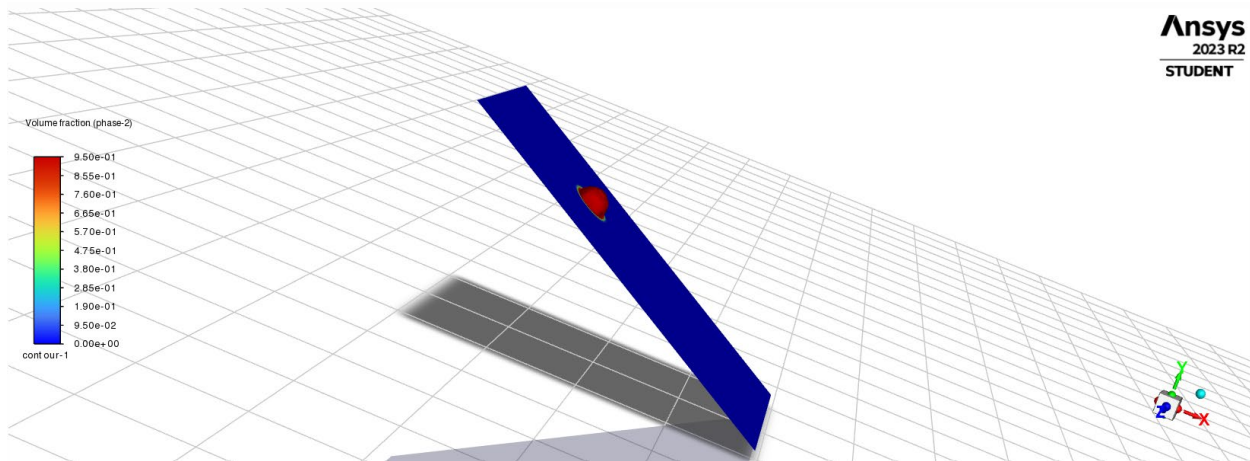


Figure 8: blob of engine oil at  $t = 0$ s

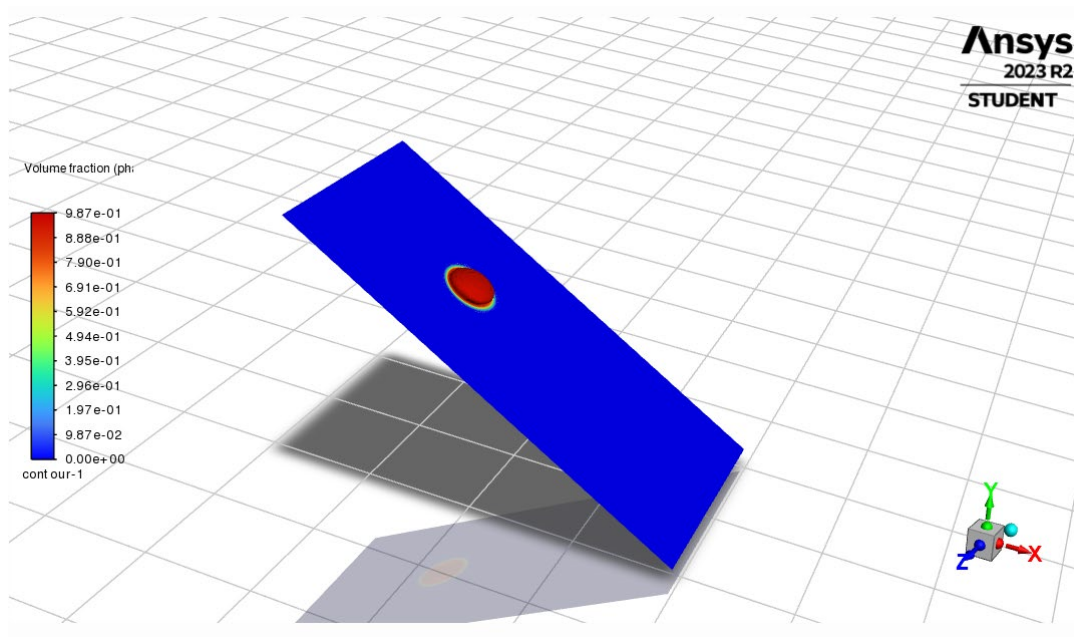


Figure 9: blob of engine oil at  $t = 0.05$ s

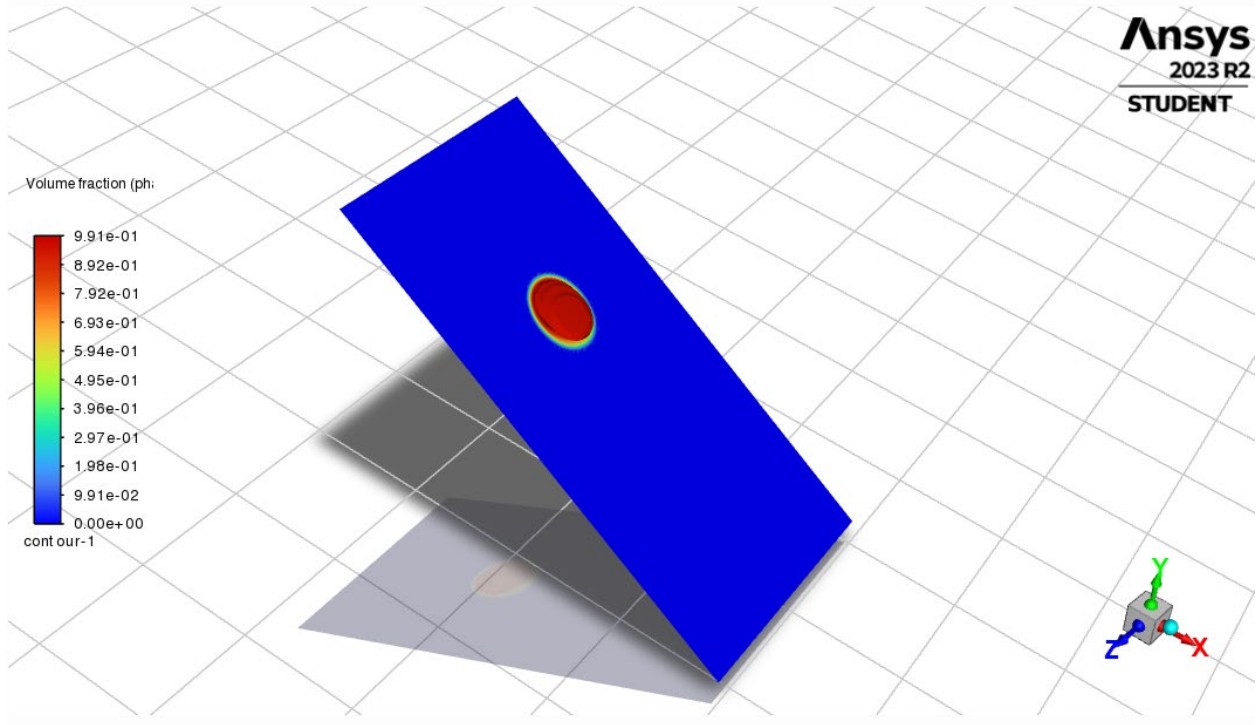


Figure 10: blob of engine oil at  $t = 0.1s$

(D8) Three contour plots of the volume fraction of engine oil on the plane of symmetry, at  $t = 0$ ,  $0.05$  s, and  $0.1$  s.

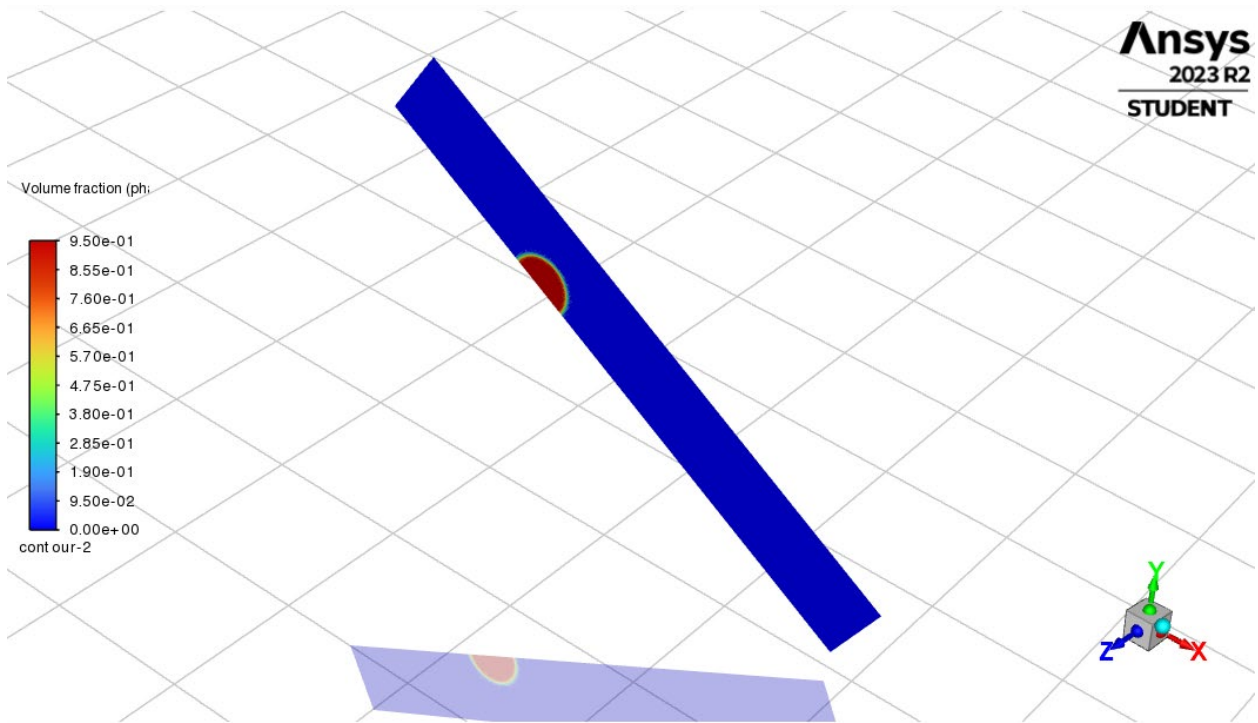


Figure 11: volume fraction of engine oil on the plane of symmetry, at  $t = 0s$



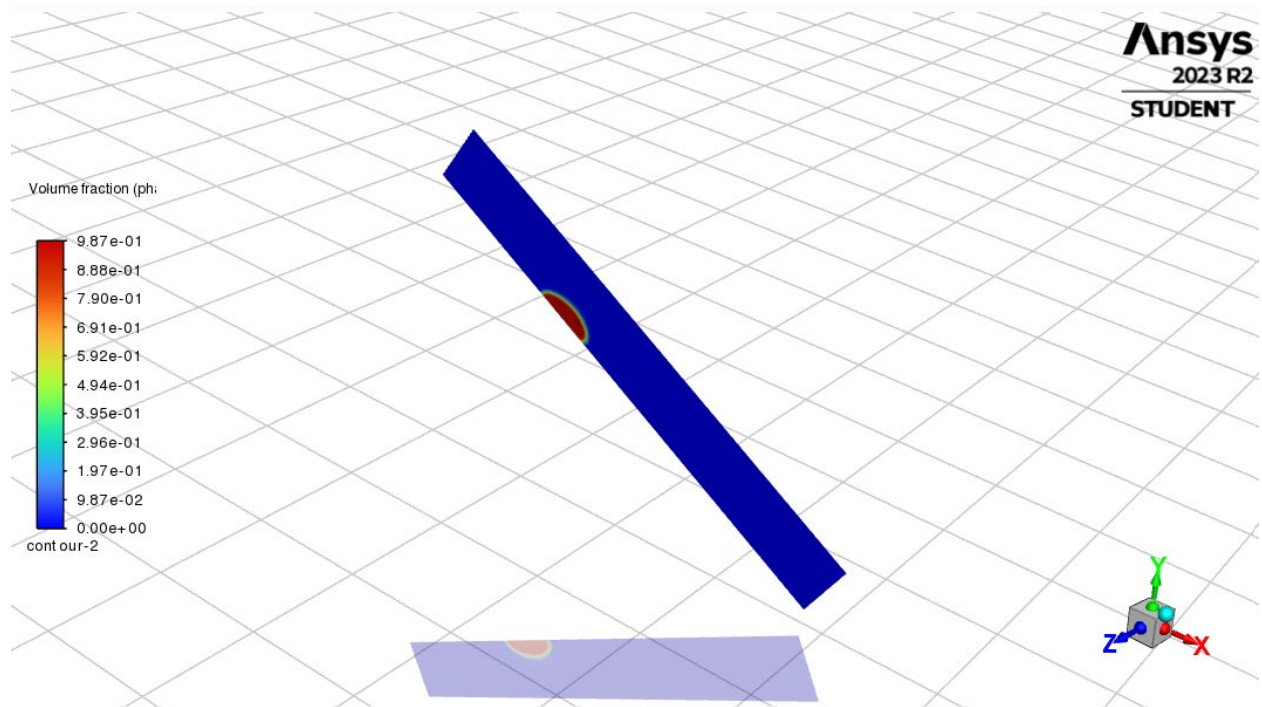


Figure 12: volume fraction of engine oil on the plane of symmetry, at  $t = 0.05s$

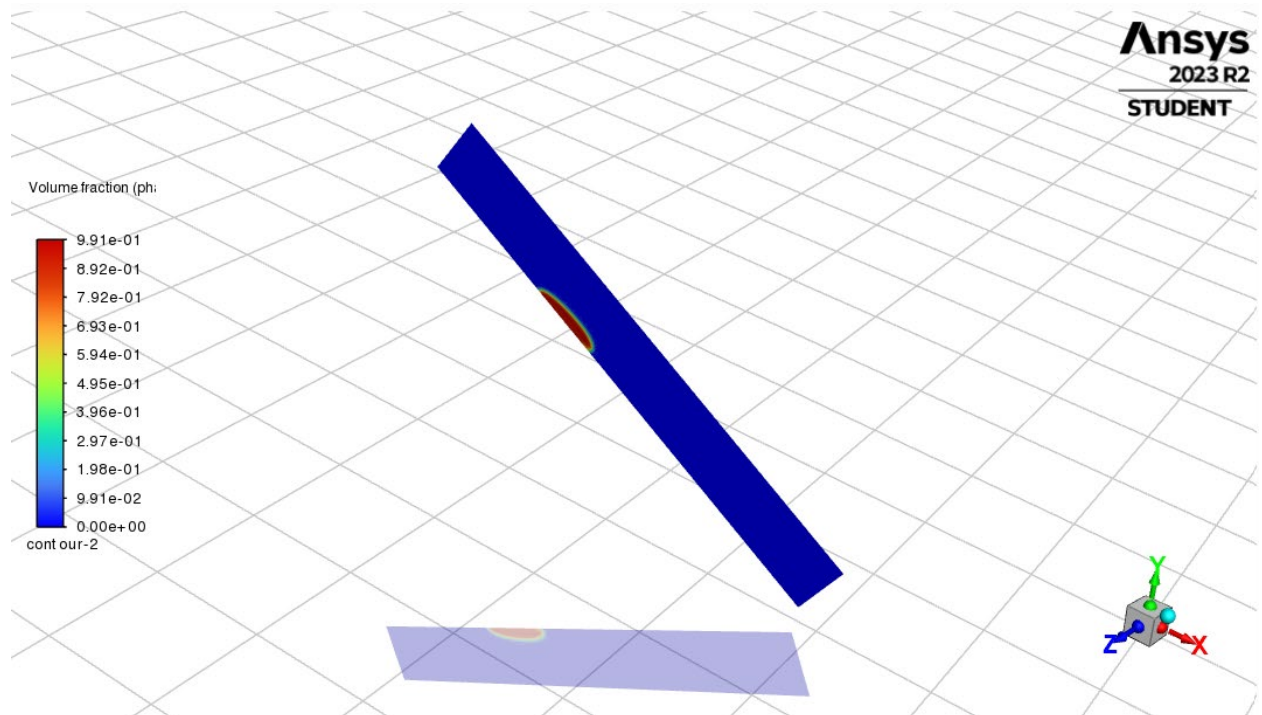


Figure 13: volume fraction of engine oil on the plane of symmetry, at  $t = 0.1s$

b) Use mercury as the liquid

(D9) Three plots in the fashion of Fig. 3b that show the 3-D shape of the blob of mercury at  $t = 0$ ,  $0.05$  s, and  $0.1$  s. See (D7) for further explanations on the format.

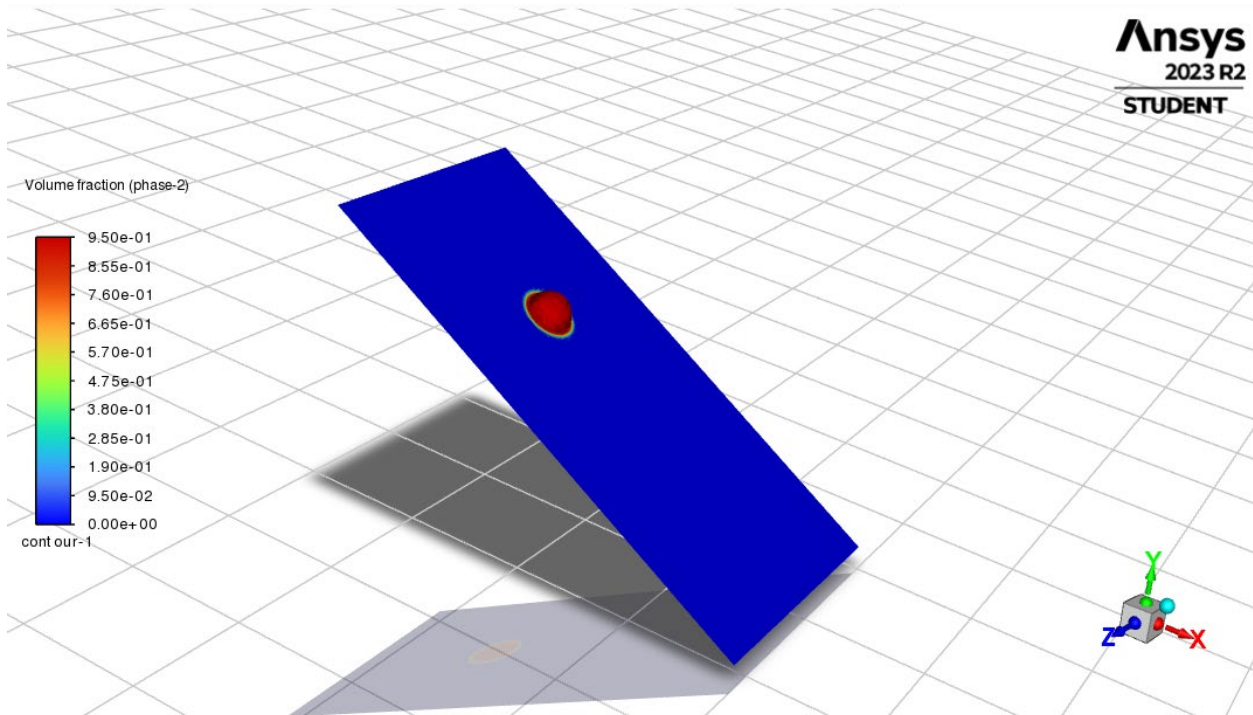


Figure 14: The blob of mercury at  $t = 0$ s

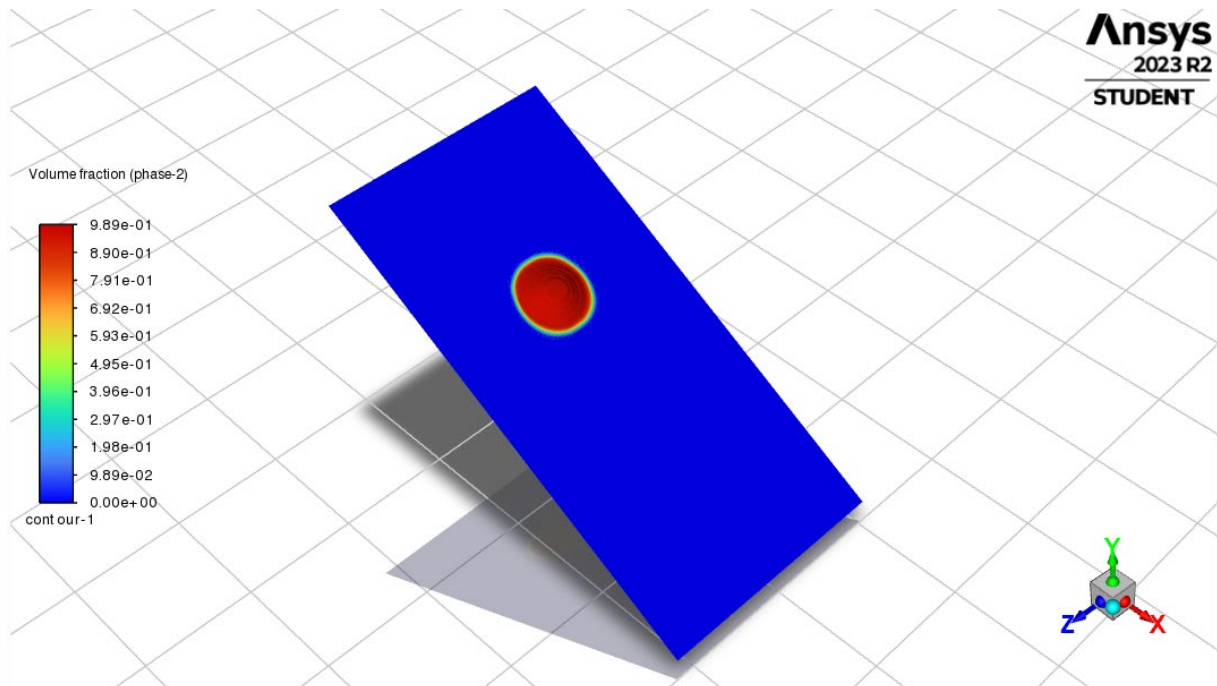


Figure 15: The blob of mercury at  $t = 0.05$ s

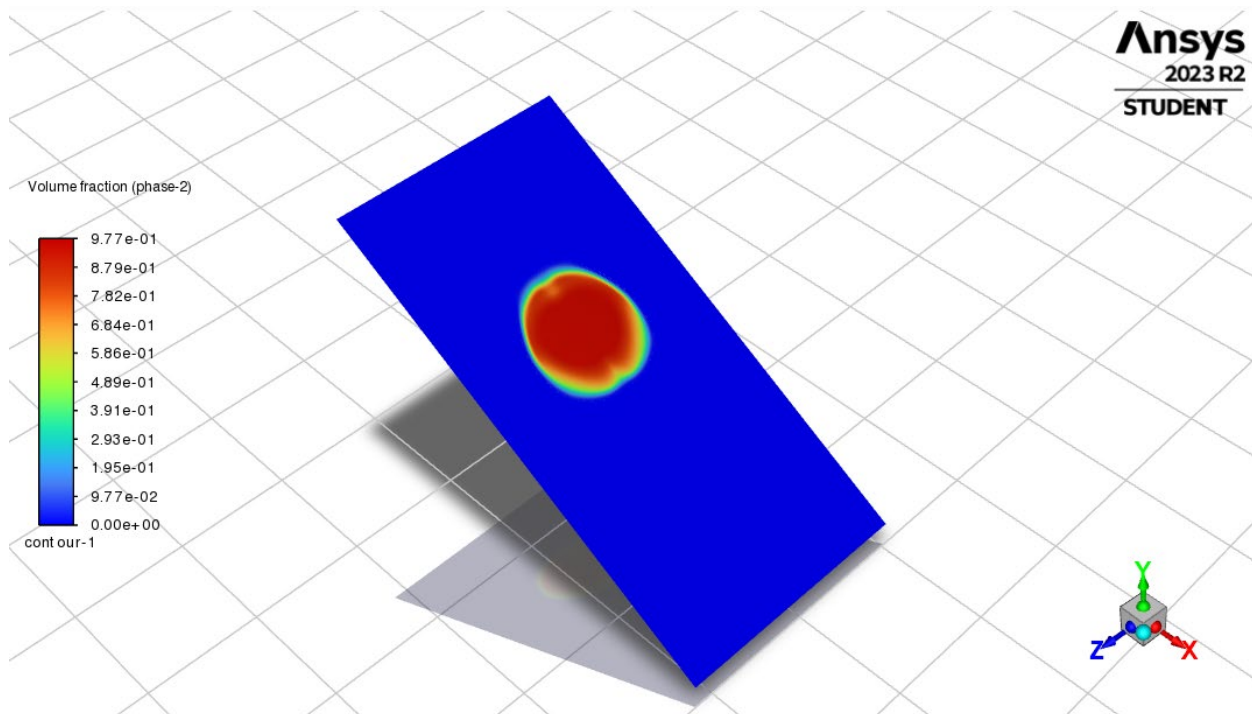


Figure 16: The blob of mercury at  $t = 0.1s$

(D10) Three contour plots of the volume fraction of mercury on the plane of symmetry, at  $t = 0$ ,  $0.05\text{ s}$ , and  $0.1\text{ s}$

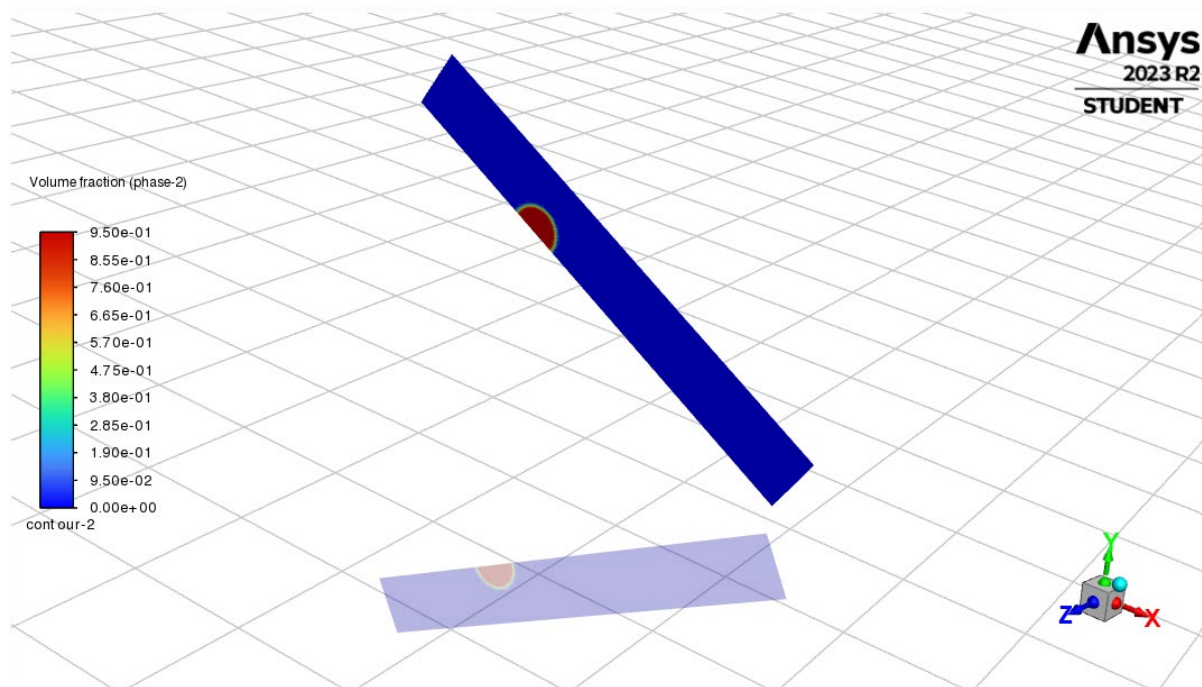


Figure 17: The plane of symmetry, at  $t = 0s$



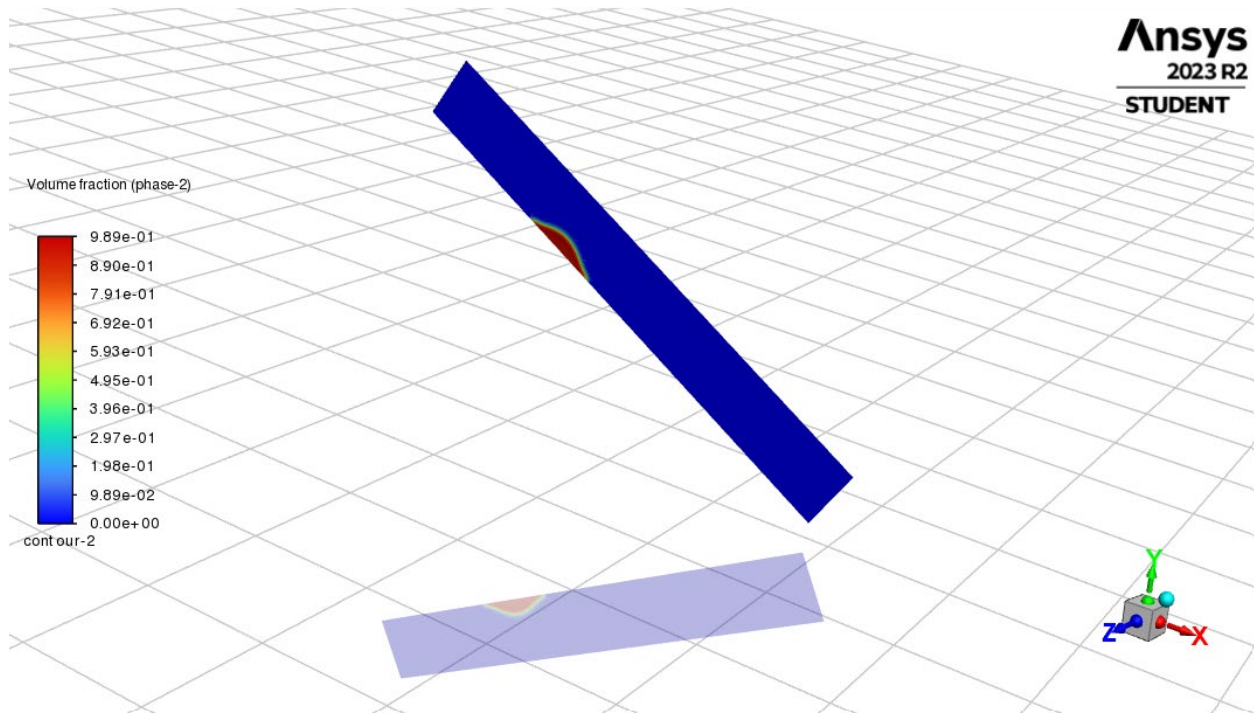


Figure 18: The plane of symmetry, at  $t = 0.05s$

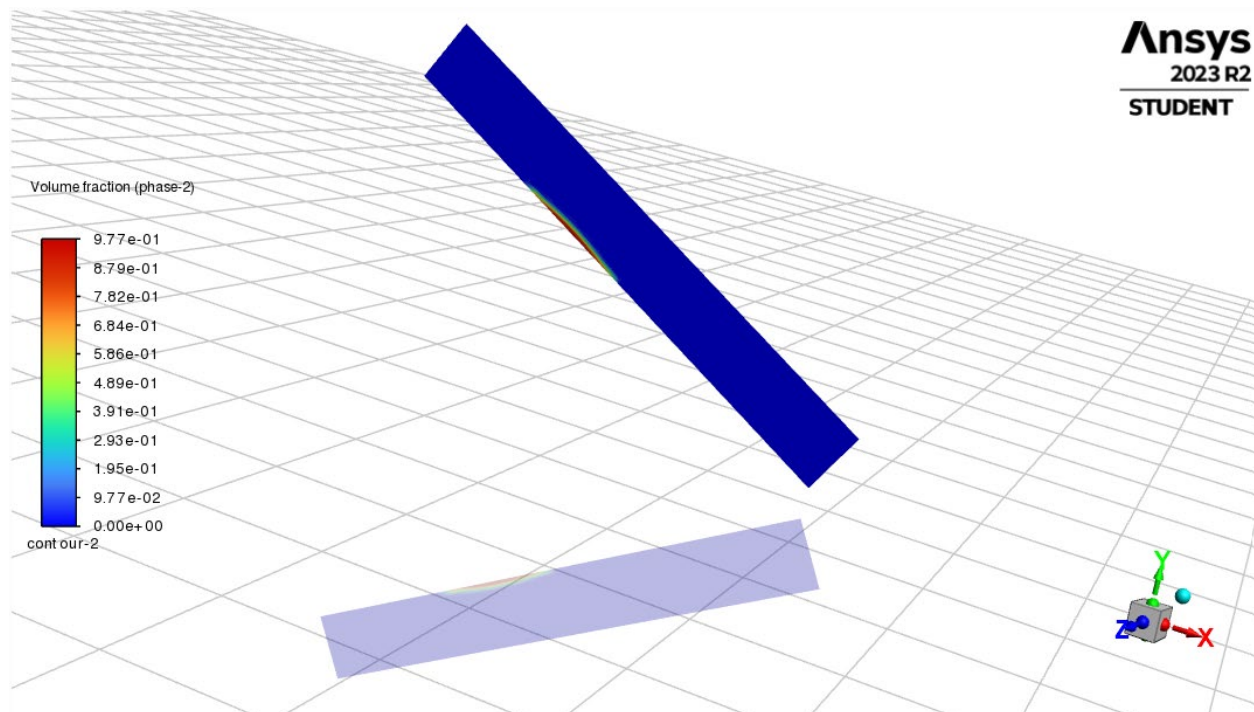


Figure 19: The plane of symmetry, at  $t = 0.1s$

(D11) A description of the computational domain, boundary conditions, mesh resolution, and time step size used in the two simulations. If different settings are used for (a) and (b), please clearly state the differences.

Mesh Resolution: 0.005 m

No. of Time Step Size: 100

Time iteration: 20

Time Step Size: 0.001

### Task 3

(D12) A description of the boundary conditions you choose for the two top openings that allow Fluent to properly simulate the oscillation. In addition, a description of the mesh resolution and time step size used for the simulation.

For the boundary the water backflow is taken as 1 which will allow the fluent to properly simulate in the oscillation.

Mesh resolution: 0.001m

No. of Time step: 1300

Time step size: 0.001s

Max. Iteration: 4

Note: As my software was crashing during simulation several times and I was running out of time, so I had to use less iterations.

(D13) A plot of the mass flow rate (in kg/s) of air at opening A (at top of the left pipe; see Fig. 5) as a function of time. The plot should cover the time to at least  $t = \tau$ . In addition, state the value of  $\tau$  (the length of one period, in second). Round it to the first digit. (For example, 2.3 s, 5.6 s. If the period is precisely 2 seconds, write it as 2.0 s.)

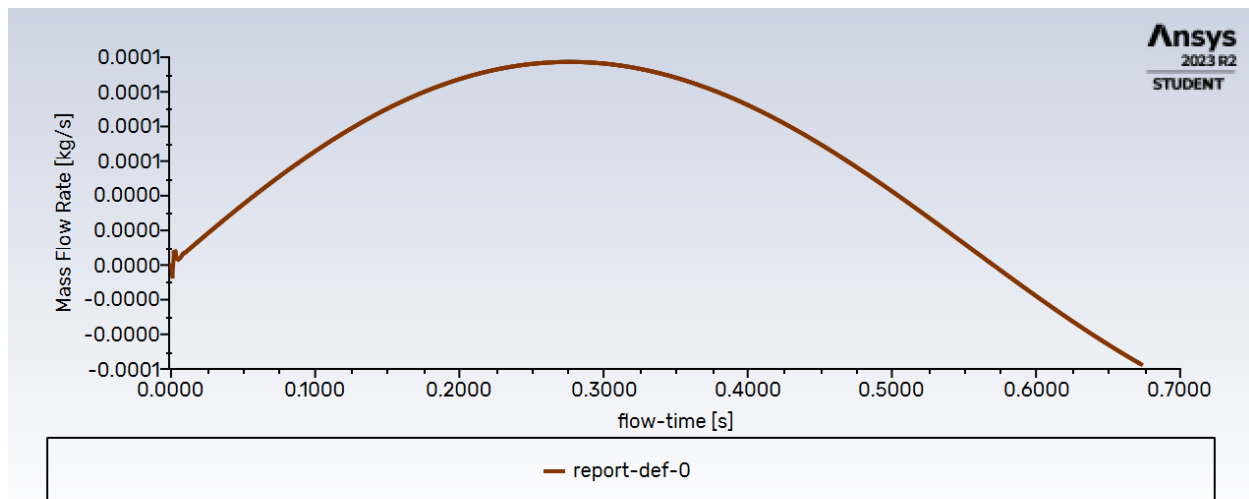


Figure 20: ) A plot of the mass flow rate (in kg/s) of air at opening A as a function of time

(D14) Contour plots of the y-velocity (see Fig. 5a for the definition of the y-direction) of the mixture on the plane of symmetry, at  $t = 0.25 \tau$  and  $t = 0.75 \tau$ .

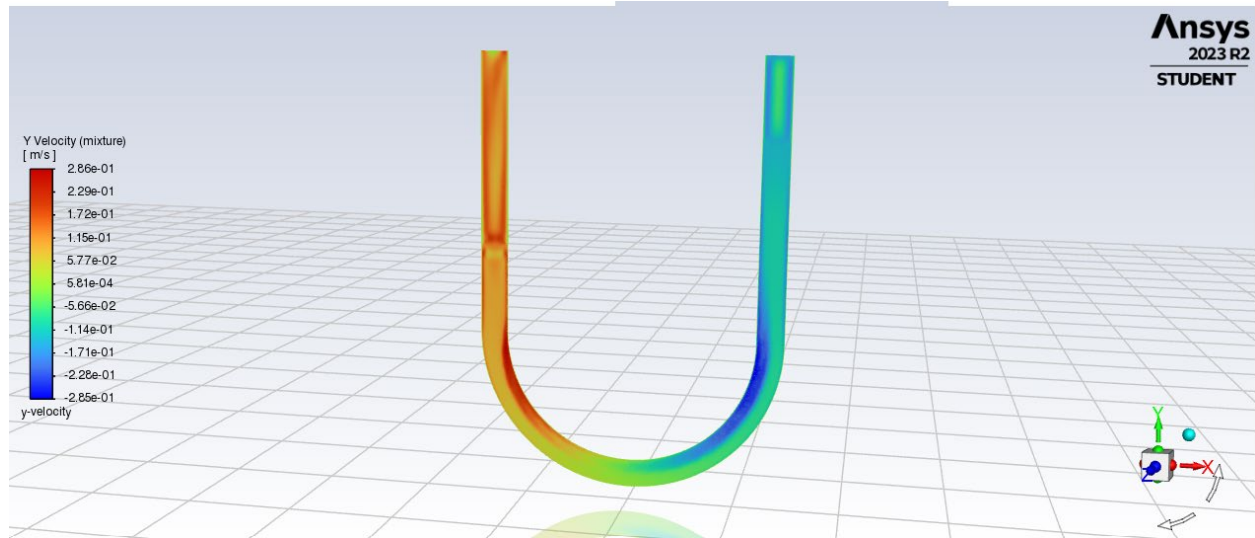


Figure 21: Contour plots of the y-velocity