Statement of Collaboration

Author: Avinash Raj DGSK
ID: 1220420240
No Collaboration

Task – 1: Simulation of leaking of natural gas from an underground vault

Case -1
The simulation of leaking of natural gas from an underground vault into open air is conducted in a 2D system. The underground vault has its lateral and top boundaries open to air as shown in the meshing figure. The smaller side is connected to a pressurized reservoir of methane. Pressure based solved was used including gravity et to -9.81m/s along the y-axis. The underground vault's lateral and top boundaries are set to pressure outlets with zero gauge pressure and with backflow phase of phase 2 (methane) set to 0; setting the side pipe's inlet to pressure inlet with gauge pressure = 75 Pa and with a volume fraction for phase 2 (methane) set to 1; initializing the system with gauge pressure = 0 Pa, x and y velocity = 0 m/s, phase 2 (methane) volume fraction set to 0, turbulence kinetic energy to $1 \times 10^{-5}\, m^2 s^{-2}$, and turbulence dissipation rate to $1 \times 10^{-6}\, m^2 s^{-3}$. This transient simulation was performed for t = 5s.

<u>D1- Contour plots of the volume fraction of methane, y-velocity and static pressure. (t = 5 s)</u>

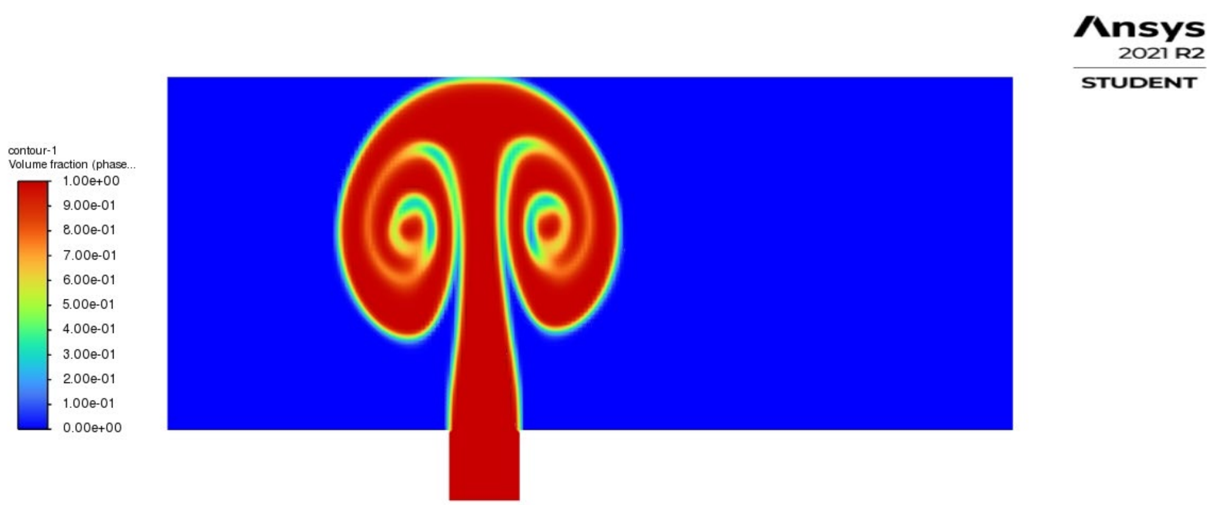


Figure 1: Volume fraction of Methane

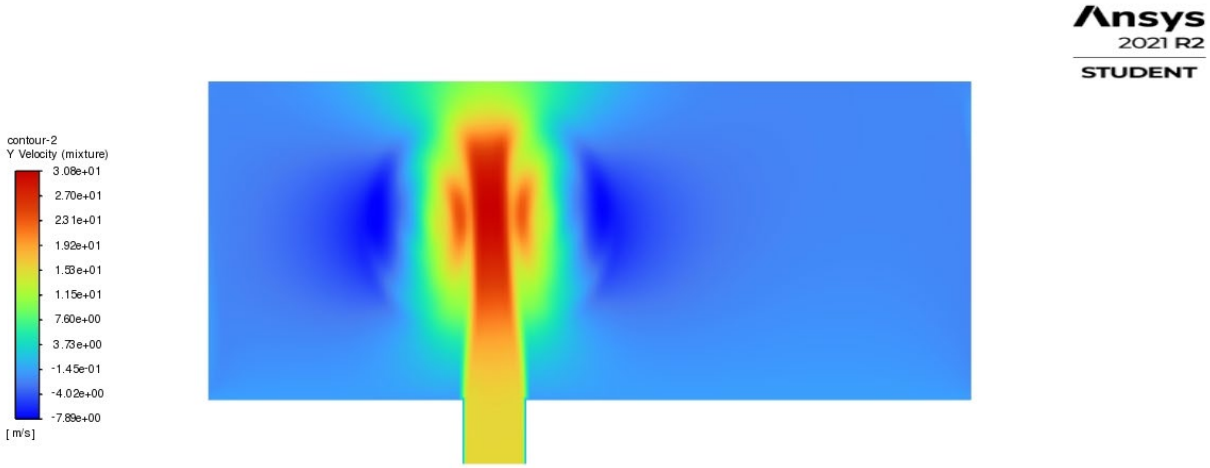
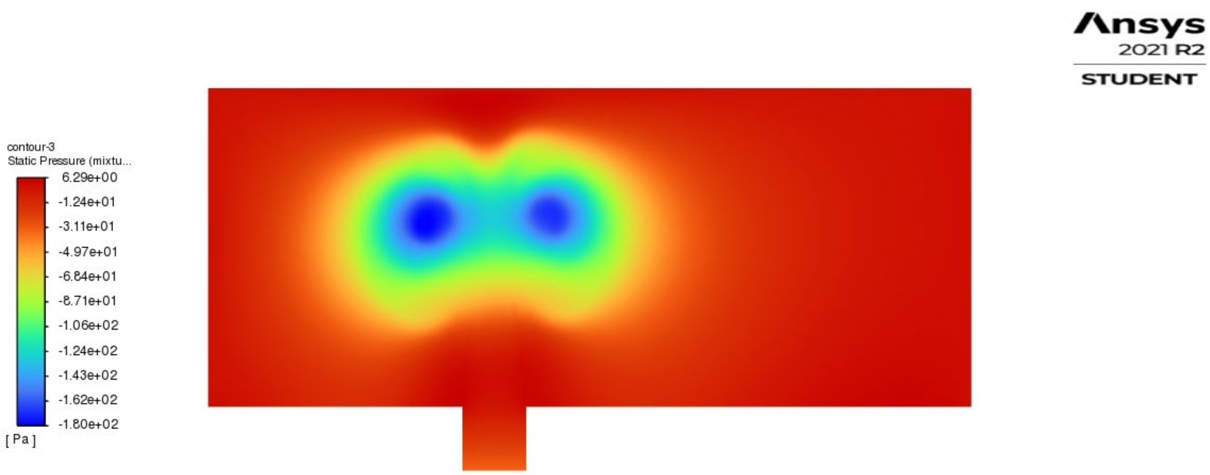


Figure 2: y-velocity



Number of Time step	Step Size	Maximum iterations
250	0.02	20

Table 1: Transient solution parameters

Element Size	Number of elements	Number of nodes
0.3m	16928	17229

Table 2: Mesh parameters

Case - 2

This case is carried out that of the above. However, the left lateral side is set to a velocity inlet with an imposed velocity profile set to $u = 0.4y - 0.008y^2$ (m/s) with the pressure still set to 0 Pa and a volume fraction for phase 2 (methane) set to 0.

D2- Contour plots of the volume fraction of methane (t = 5 s)

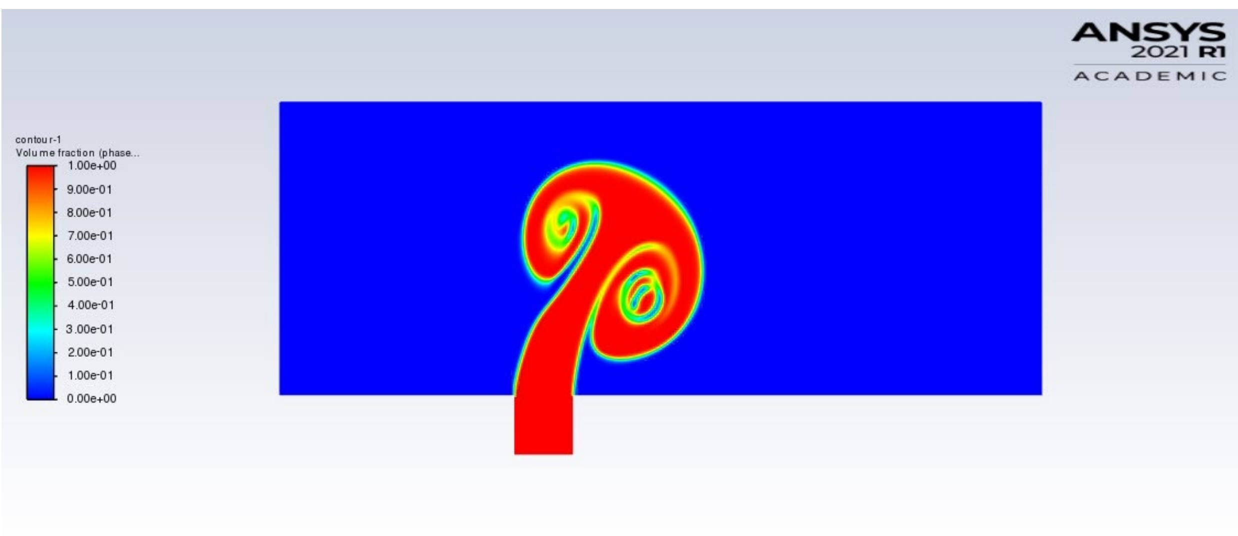


Figure 4: volume fraction of methane

D3- Mesh resolution and time step size

Number of time steps	Step Size	Maximum iterations
250	0.02	15

Table 3: Transient solution parameters

Element Size	Number of elements	Number of nodes
0.2m	41250	41726

Table 4: Mesh parameters

Task – 2 : Simulation of impact of water droplet on a flat water surface.

This task simulates the process of a falling water droplet impacting on a flat-water surface, in a pure 2-D setting. This simple 50 cm x 50 cm square bucket is open to air at the top with the other three sides being walls. A quad region was created to partially fill water to a depth of 15 cm in the square bucket, and a circular region of water, with a diameter of 6 cm, is placed 40 cm above the water surface. Running a transient simulation and using a pressure-based solver, this simulation included turning on gravity, g = -9.81 m/s^2 .The vertical "y direction"; using a laminar model; turning on surface tension (0.0719404 N/m); setting the square bucket's top boundary to a pressure outlet with zero gauge pressure and with backflow phase of phase 2 (water) set to 0; initializing the system with gauge pressure = 0 Pa, x and y velocity = 0 m/s, and phase 2 (water) volume fraction to 0. This transient simulation was performed for t = 0.3 s.

D4- Contour plots of the volume fraction of water

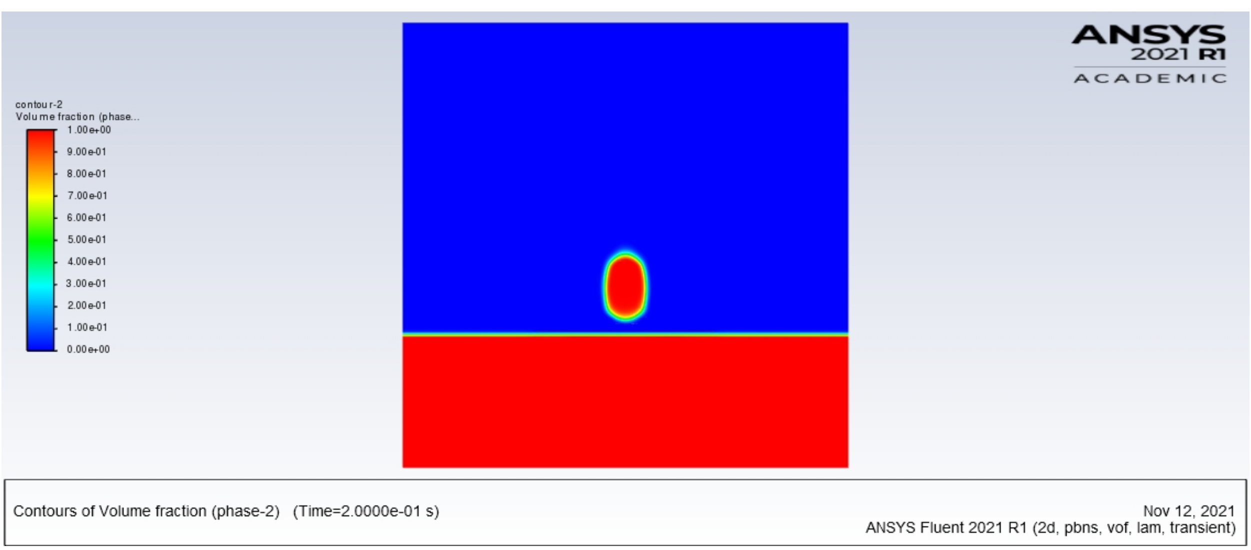


Figure 5: Volume fraction of water at t=0.2s

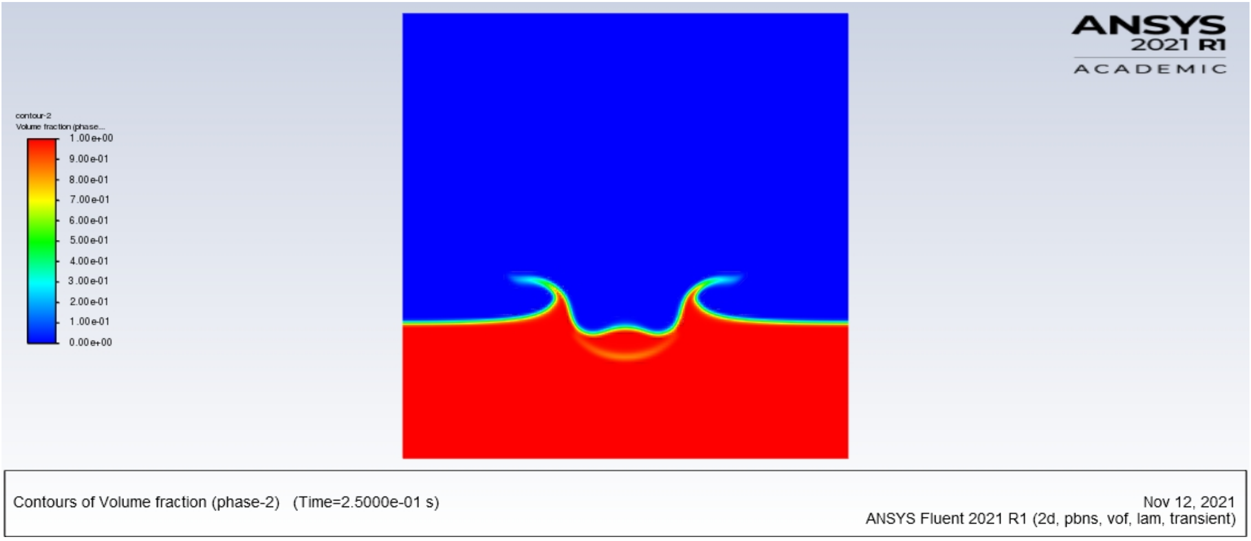


Figure 6: Volume fraction of water at t=0.25s

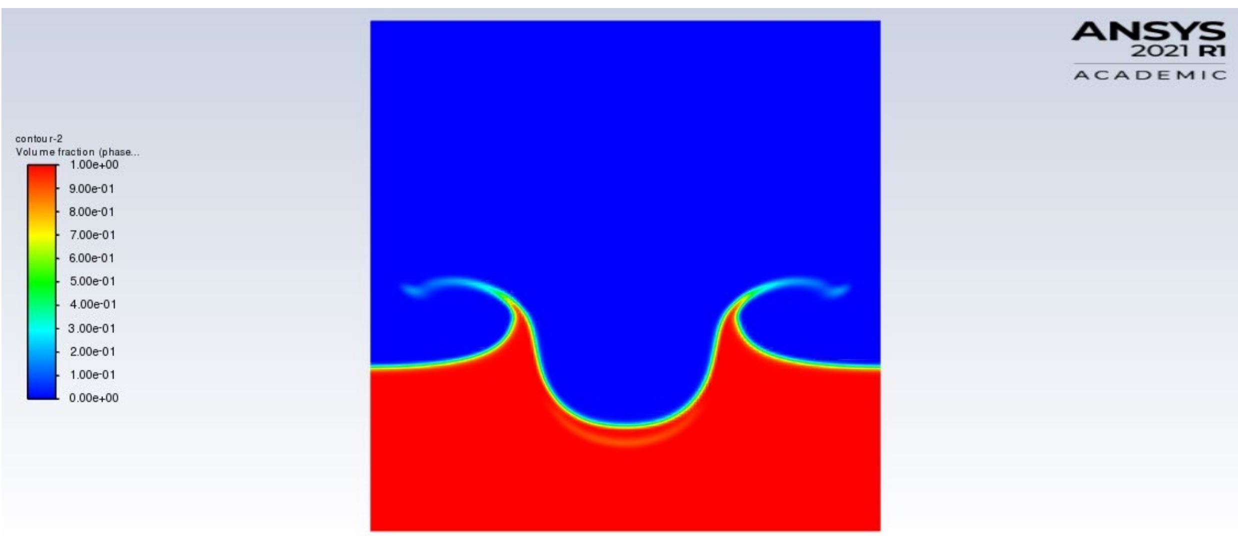


Figure 7: Volume fraction of water at t=0.3s

D5- Mesh Resolution and Time Step used in the solution

Element Size	Number of elements	Number of nodes
0.003m	27889	28224

Table 2: Mesh parameters

Number of time steps	Step size	Maximum iterations
t=0.2s, 200	0.001	20
t=0.25s, 250	0.001	20
t=0.3s, 250	0.0012	20

Table 3: Transient solution parameters

Task – 3: Simulation of engine oil droplet on an inclined surface

This task simulates the temporal evolution of engine oil droplet on a 45° inclined plate, in a 3-D system. This 45° inclined plate is surrounded by open air. A hemisphere region, with a radius of 3 cm, of glycerin (phase 2) was placed on the 45° inclined plate with the 45° inclined plate not including top or side boundaries. Running a transient simulation and using a pressure-based solver, this simulation included turning on gravity and setting it equal to the vertical "y direction"; using a laminar model; turning on surface tension and setting it equal to 0.03 N/m; initializing the system with gauge pressure = 0 Pa, x and y velocity = 0 m/s, and phase 2 (engine oil) volume fraction to 0. This transient simulation was performed for $t = 0.1s$.

D6- Description of computational domain, boundary conditions, mesh resolution and time step size

Number of time steps	Step size	Maximum iterations
t=0s, 100	0.001	15
t=0.05s, 100	0.0005	15
t=0.1s, 100	0.001	15

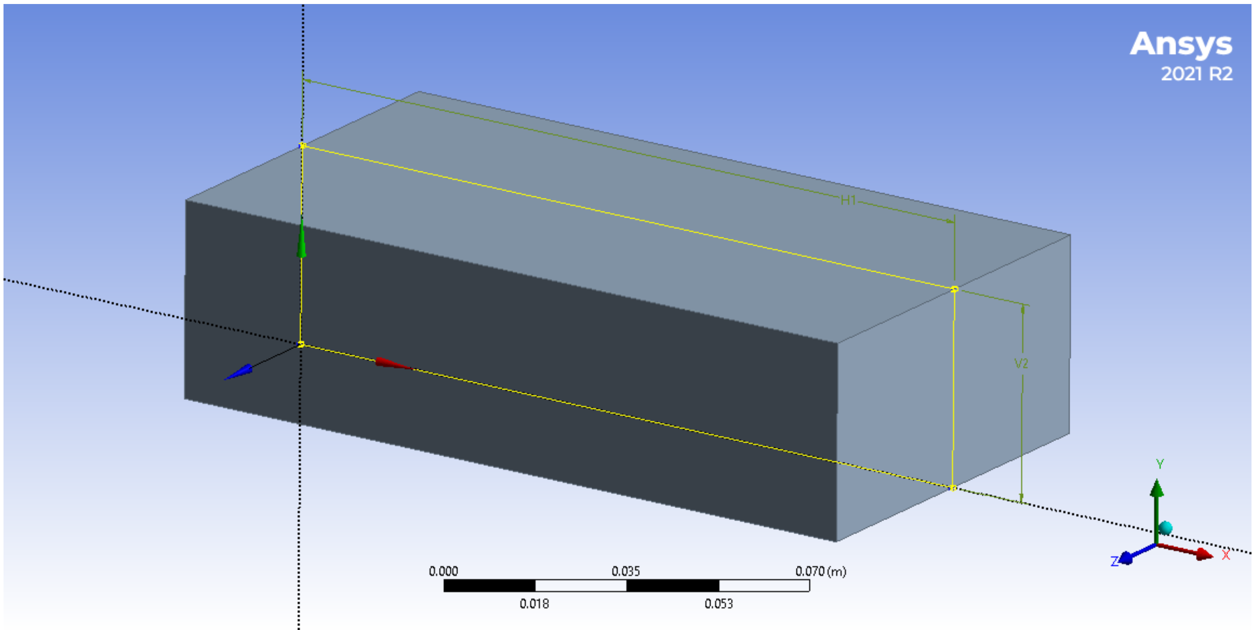
Table 3: Transient solution parameters

Element Size	Number of elements	Number of nodes
0.001m	480000	501471

Table 4: Mesh parameters

Computational Domain:

The Computational Domian is the rectangular box created by the dimensions of 10mm in X-axis, 6mm in Y-axis and a symmetric distance of 0.3mm along the Z-axis.



Boundary Conditions:

As the plate was built on flat surface, to create the inclination gravity was set to g_x= 9.81(sin(45)) and g_y= -9.81(cos(45))

<u>D7- 3-D shape of the blob of engine oil at t = 0, 0.05 s, and 0.1 s</u>

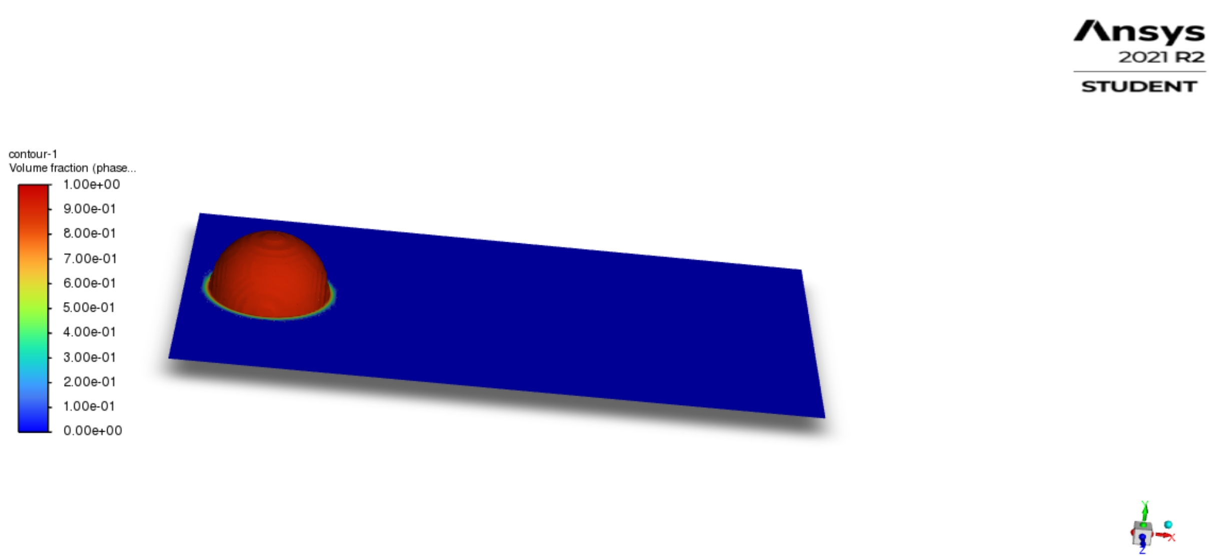


Figure 8: Visualization of engine oil droplet at t=0s

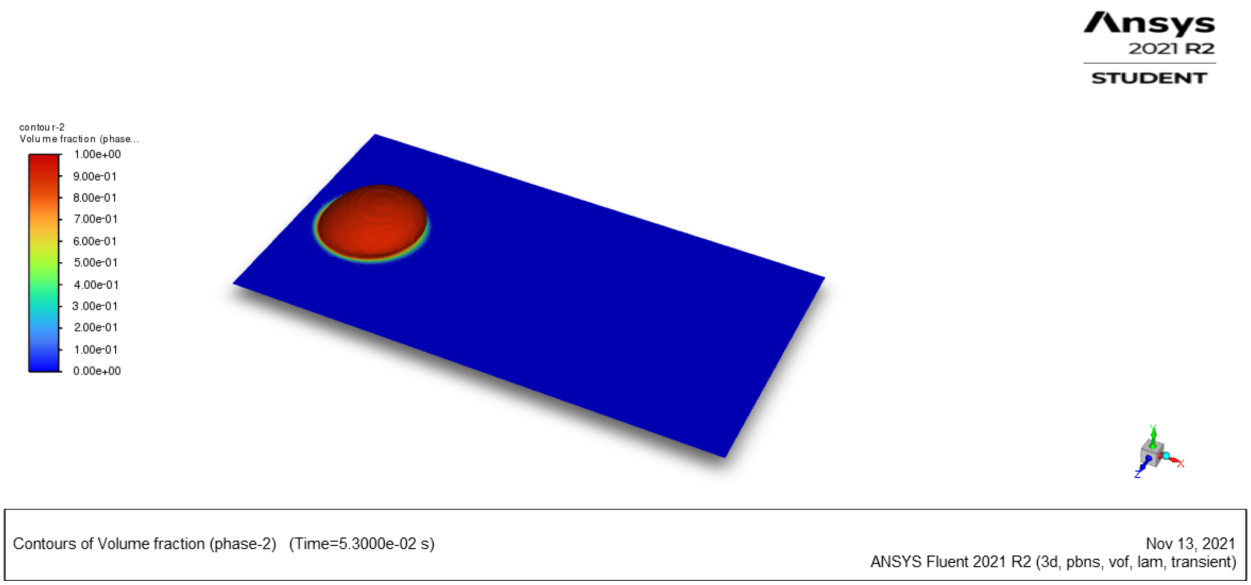


Figure 9: Visualization of engine oil droplet at t=0.05s

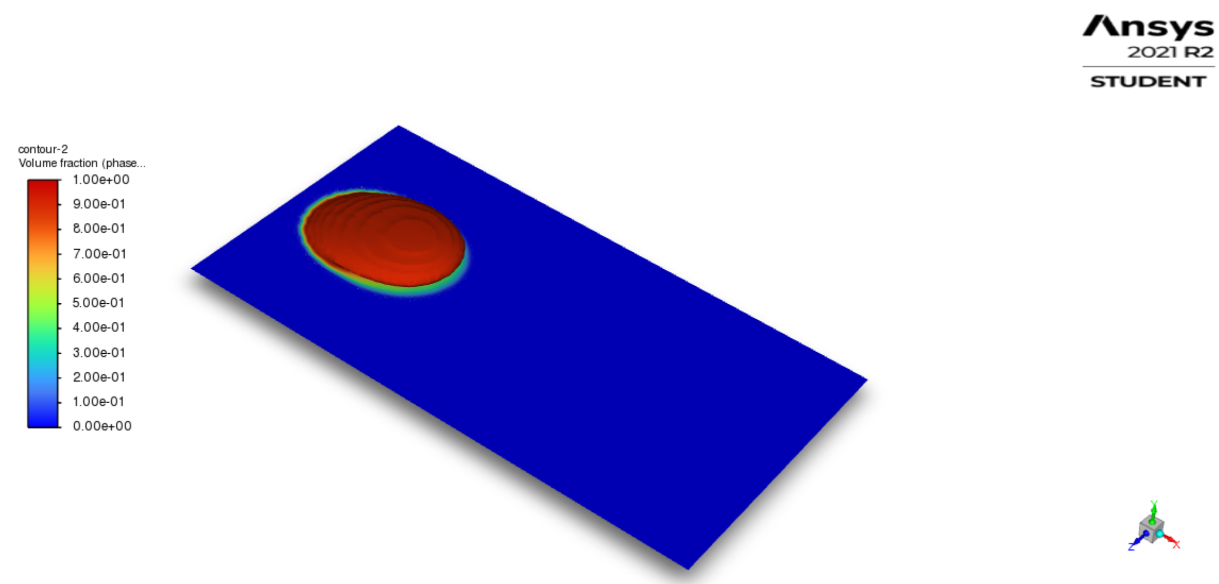


Figure 10: Visualization of engine oil droplet at t=0.1s

D8- Three contour plots of volume fraction of engine oil

Figure 11: Volume fraction of engine oil at t=0s

Figure 12: Volume fraction of water at t=0.05s

Figure 13: Volume fraction of water at t=0.1s

Task – 4 : Simulation of flow of water in U-shape 3D Tube

The purpose of this task is to simulate the oscillation of water in a u-shaped pipe with the tops open to air. I chose to use the half-pipe geometry and invoked symmetry. In this simulation, the left and right openings at the top of the pipe used boundary conditions were set to zero gauge pressure outlets with air backflow. This was done to allow the air to freely move in and out of the pipe as the water oscillates

<u>D9 Boundary conditions</u>

The purpose of this task is to simulate the oscillation of water in a u-shaped pipe with the tops open to air. I chose to use the half-pipe geometry and invoked symmetry. In this simulation, the left and right openings at the top of the pipe used boundary conditions were set to zero gauge pressure outlets with air backflow. This was done to allow the air to freely move in and out of the pipe as the water oscillates.

D10 Plot of h as a function of time

Figure 14: Line plot of h(t)

D11 x-velocity and y-velocity at the plane of symmetry

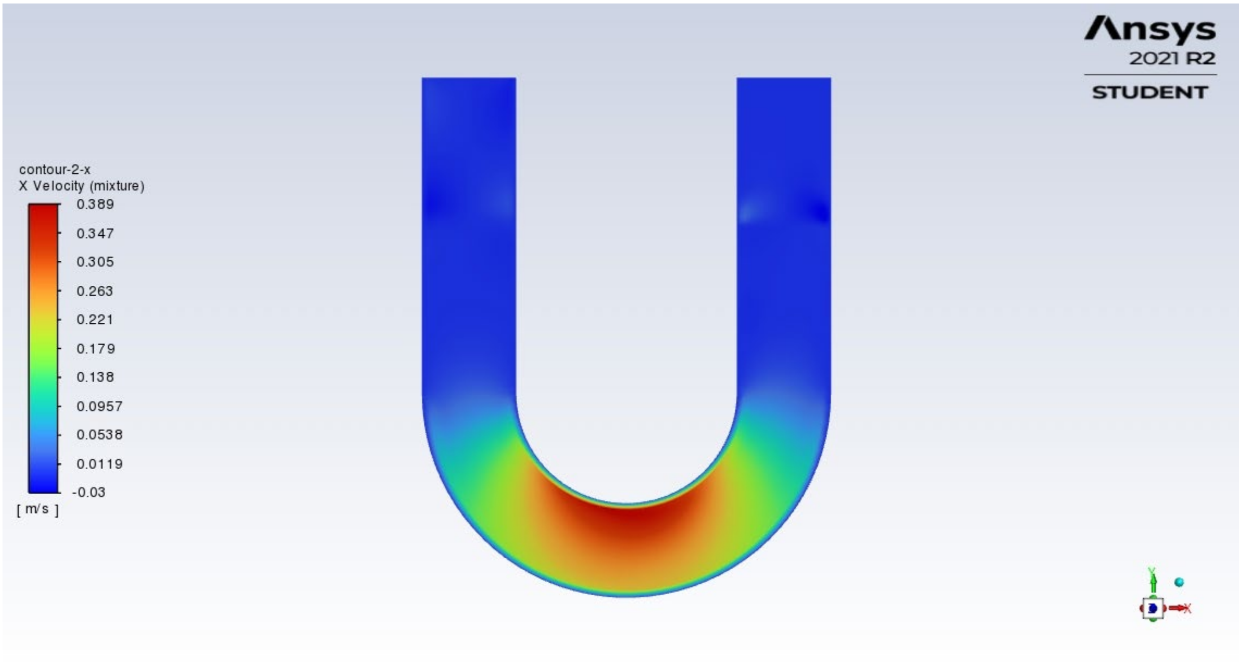


Figure 15: X-Velocity Contour plot at t1

Figure 16: y-velocity contour plot at t1