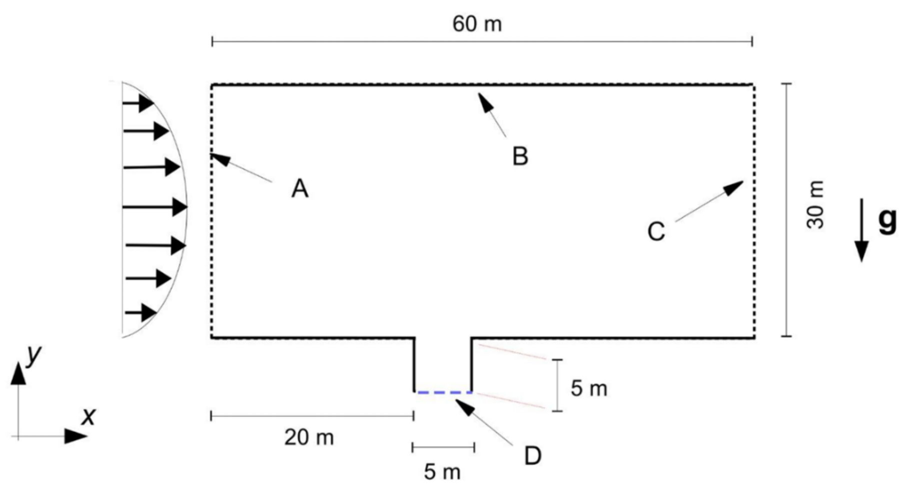
MAE 560 – Applied Computational Fluid Dynamics
Project-2

Arya Narasimha Raju
1229973903

Statement of Collaboration: “No Collaboration”

Task 1:

Task 1 is simulation of leaking of natural gas from underground vault into open air, in a 2D setting. This task includes two transient simulations. In both, use the default constant values of density and viscosity from Fluent database for air and methane, and set gravity to the regular g = –9.81 m/s2 in the vertical direction.



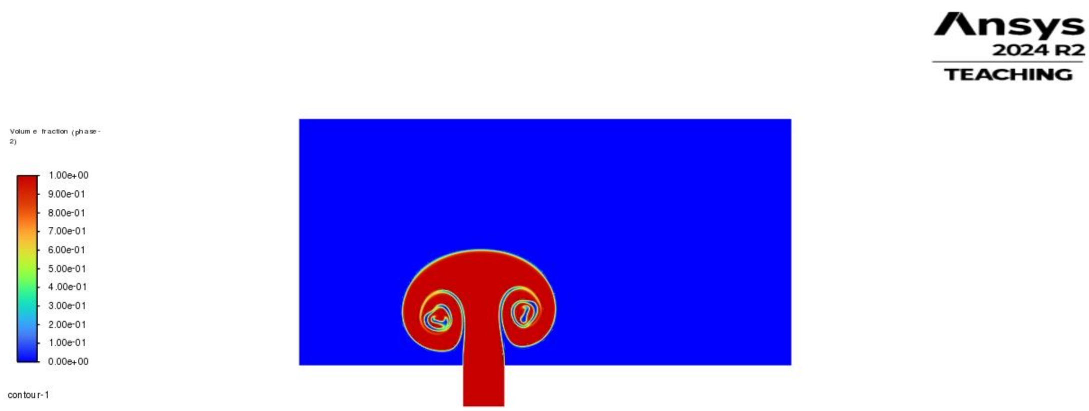
Case (a):

1. Set Outlet and Inlet of 100 Pa boundary conditions

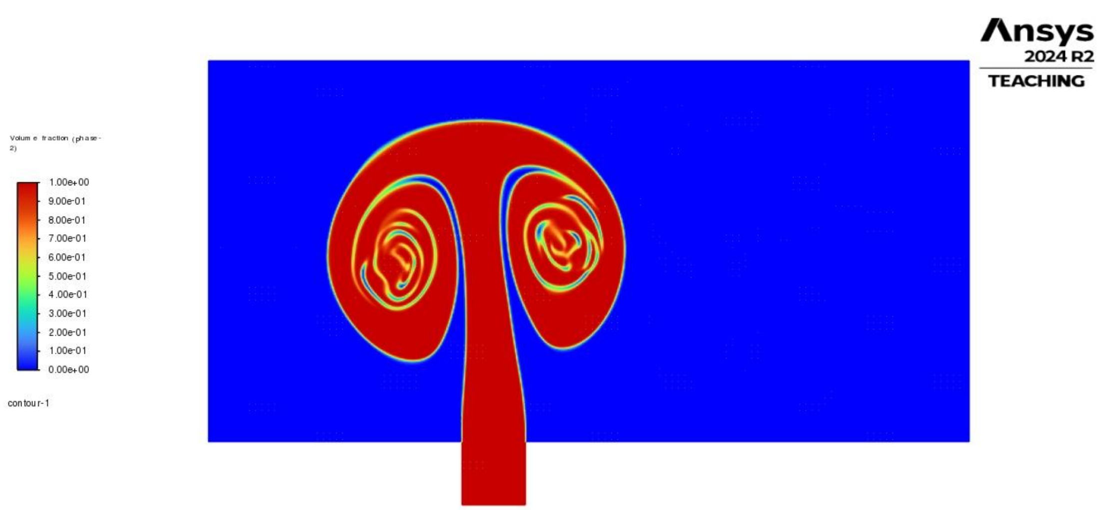
2. Initialize the system with gauge pressure = 0 and velocity = 0. For the turbulence parameters, initialize turbulence kinetic energy (k) = 1 $m2s{-}2$, and specific dissipation rate (omega) = 1 $s{-}1$. Perform a transient simulation to t = 4.5 s

(D1). Contour Plots of the *volume fraction of methane* at t = 3 s and t = 4.5 s.

Methane at *t = 3* seconds:



Methane at *t = 4.5* seconds:

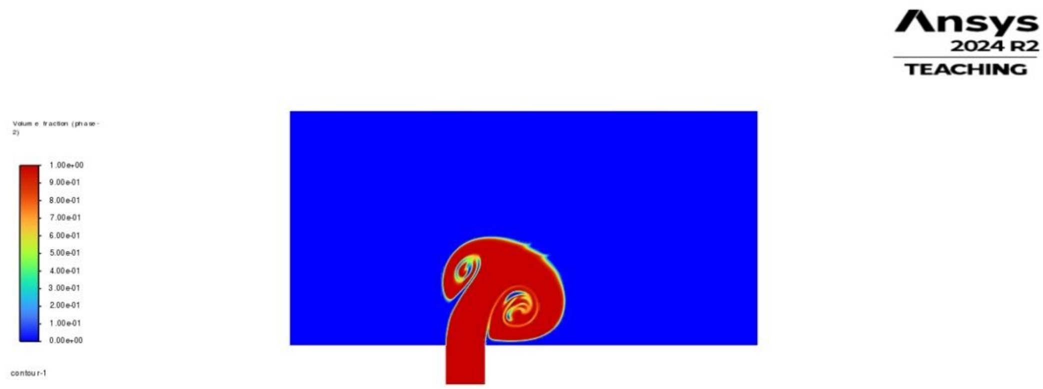


Case(b):

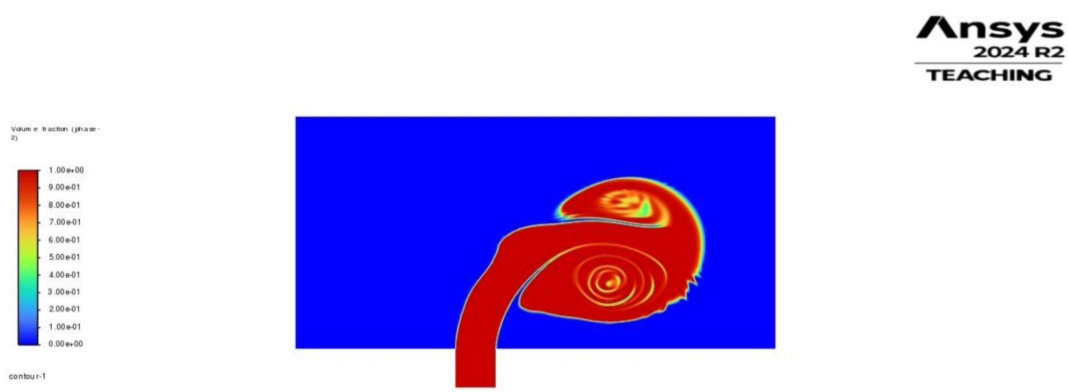
1. Use the same setting as Case A except that the gauge pressure at inlet D is increased to 250 Pa. In addition, the left boundary marked by A is replaced by a velocity inlet, with an imposed velocity profile for the x-velocity given by u = 0.6 y – 0.02 y2 where u is in m/s and y in m.

2. This gives a parabolic profile with u = 0 at the ground (y = 0) and top of the domain (y = 30 m), and u attains a maximum of 4.5 m/s at y = 15 m.

(D2). Contour plots of the *volume fraction of methane* at t = 3 s and t = 4.5 s.
Methane at *t = 3* seconds:



Methane at *t =4.5* seconds:



(D3).

Mesh Resolution:

Element Size Used: 0.01m

Statistics	
Bodies	1
Active Bodies	1
Nodes	183451
Elements	182500
Mesh Metric	None

Run Parameters:

Time Step Size: 0.01

Max Number of Iteration per time step: 20

Number of Time steps: 300

Same parameters are used for both conditions.

Task 2

An inclined plate that forms a 40° angle with the ground. At t = 0, a droplet of liquid is placed on the plate, and it is shaped like a hemisphere with a radius of 1 cm (diameter of 2 cm). The droplet is surrounded by open air. Gravity is the regular g = – 9.81 m/s2 in the vertical direction

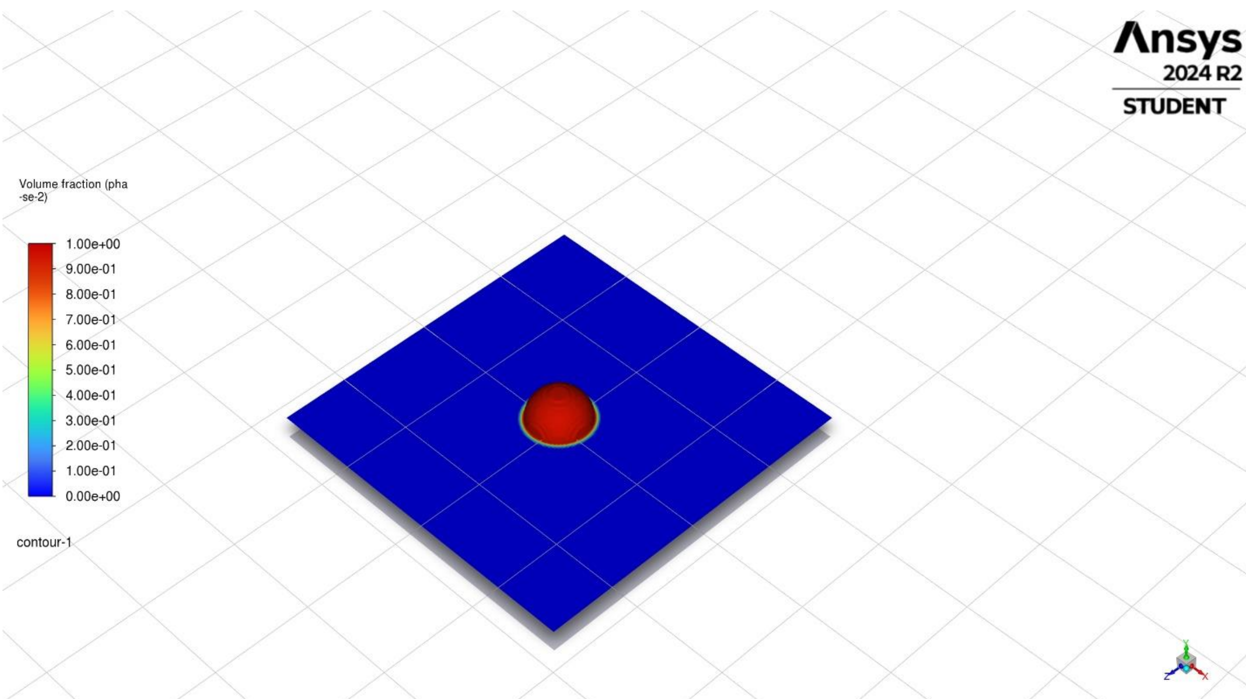
(a) Using Glycerin as liquid

Setting surface tension = 0.03 N/m.

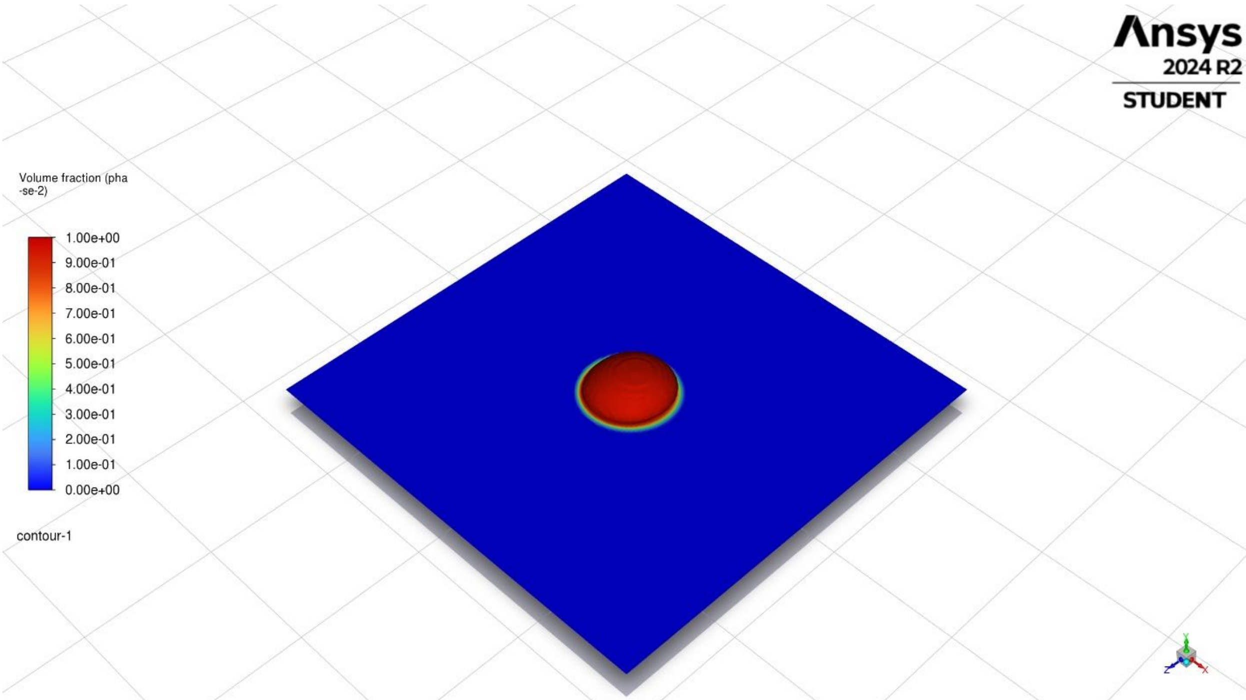
Transient solution to = 0.06s.

(D4). Three plots of 0s, 0.03s and 0.06s with VF=0.95 in fashion of Fig 2b. given in question:

t=0 s

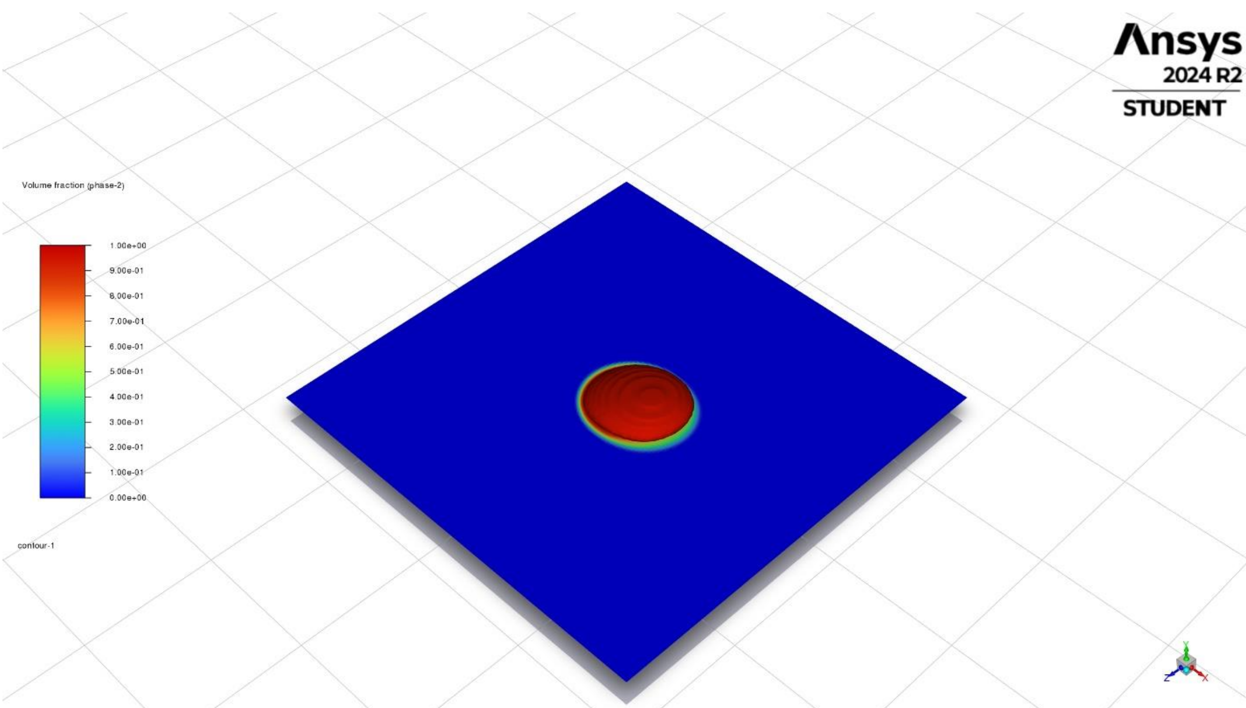


t=0.03s



Ansys
2024 R2
STUDENT
Volume fraction (pha
-se-2)
1.00e+00
9.00e-01
8.00e-01
7.00e-01
6.00e-01
5.00e-01
4.00e-01
3.00e-01
2.00e-01
1.00e-01
0.00e+00
contour-1

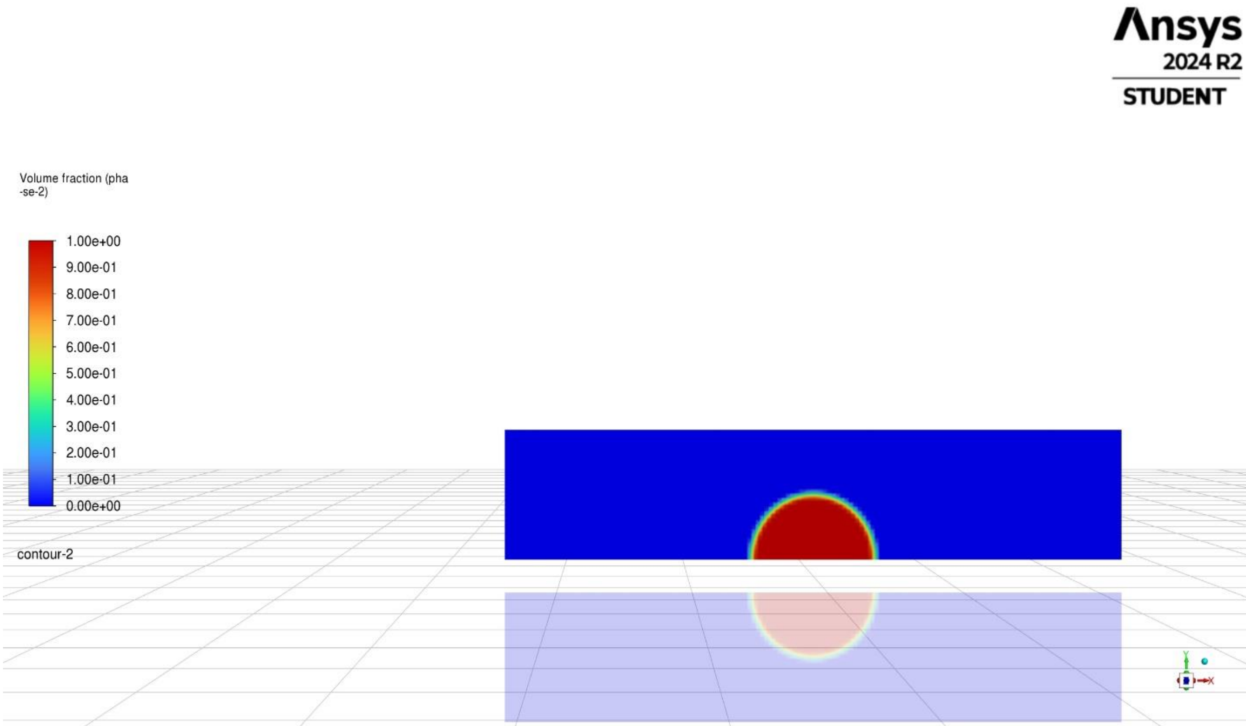
t=0.06s



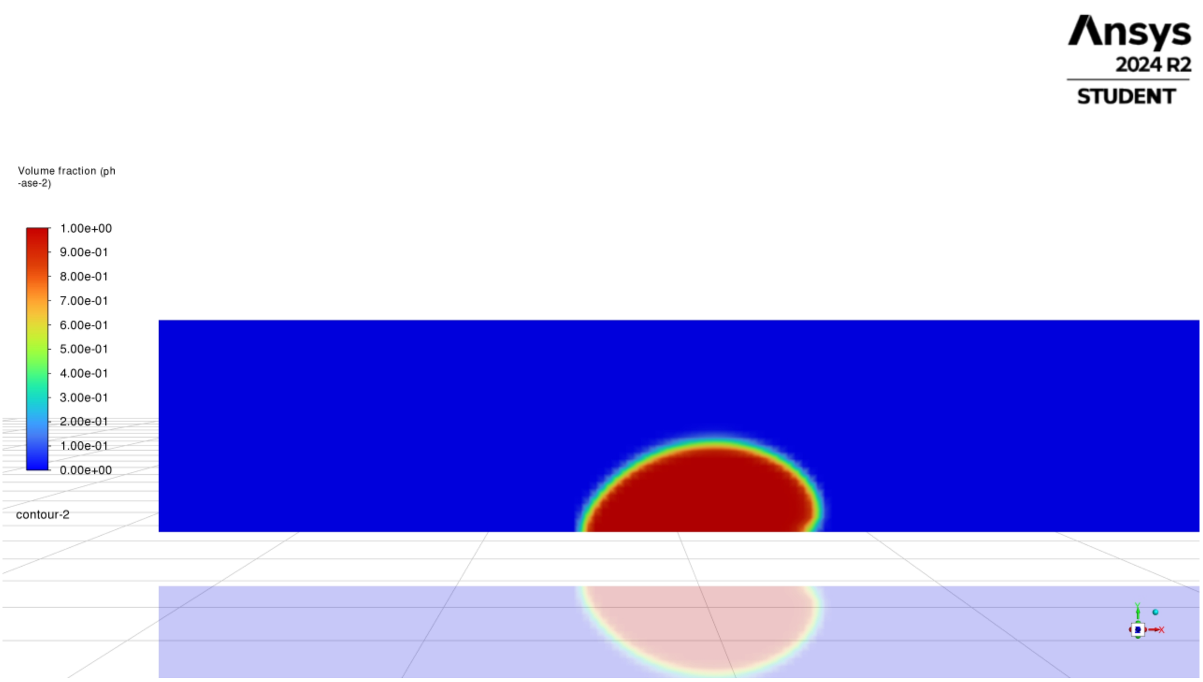
Ansys
2024 R2
STUDENT
Volume fraction (phase-2)
1.00e+00
9.00e-01
8.00e-01
7.00e-01
6.00e-01
5.00e-01
4.00e-01
3.00e-01
2.00e-01
1.00e-01
0.00e+00
contour-1

(D5). Three plots of volume fraction of glycerin on the plane of symmetry, at t=0s, 0.03s, 0.06s.

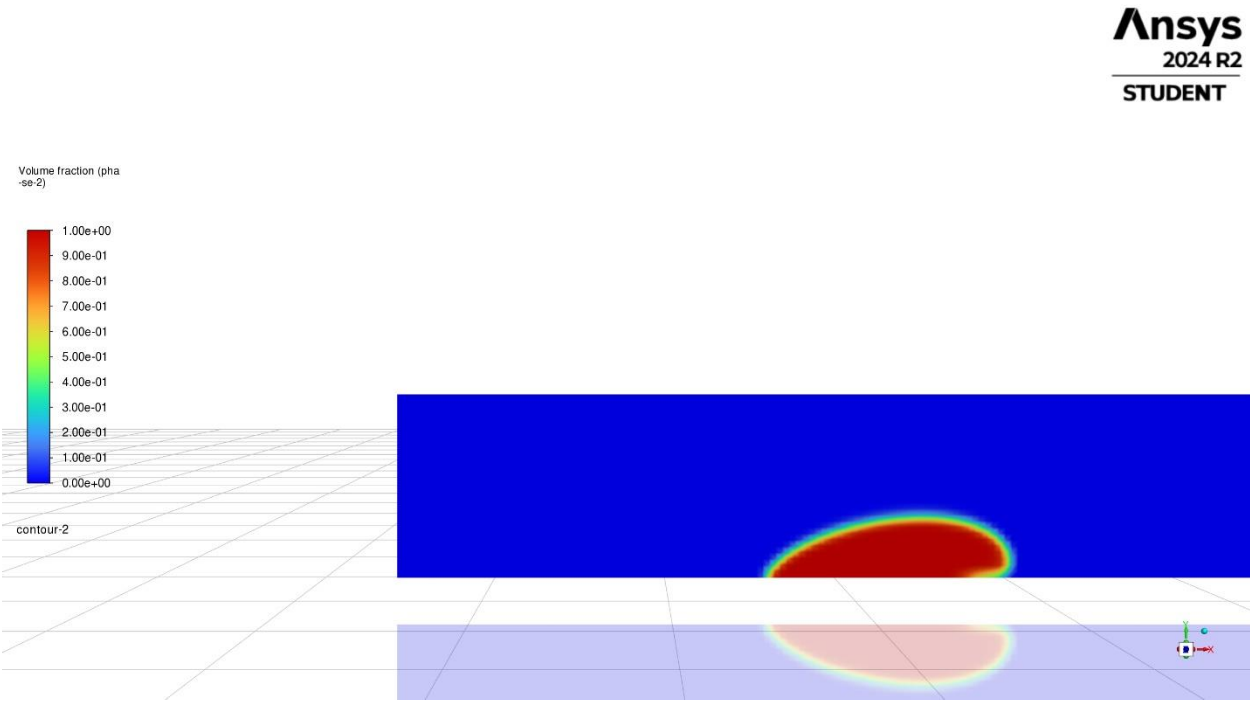
t=0s



t=0.03 s



t=0.06s

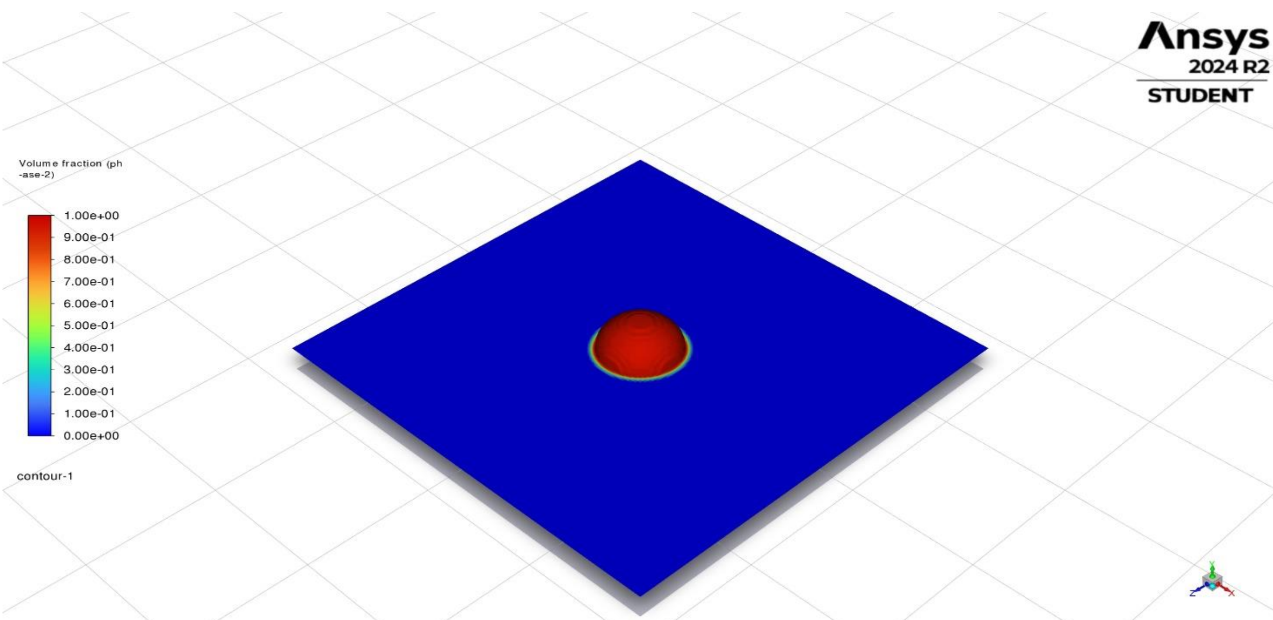


(b) Using mercury as the liquid

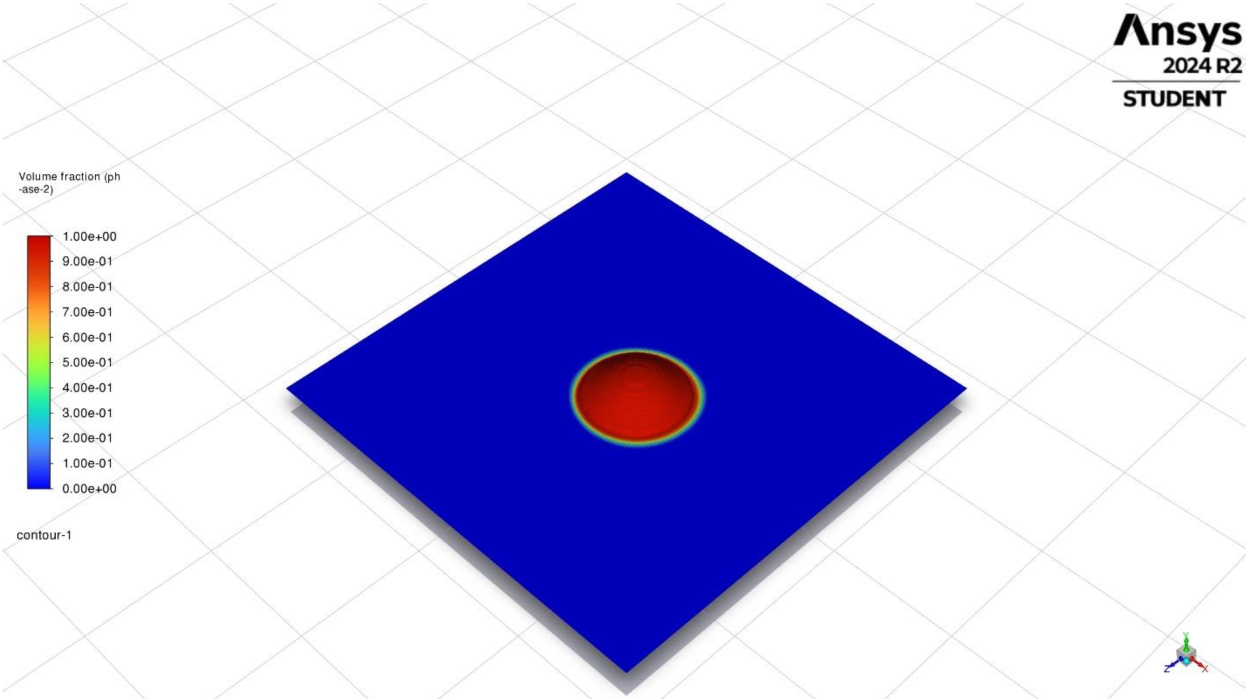
Setting coefficient = 0.5 N/m and running transient simulation to t = 0.06s.

(D6) Three plots in fashion of fig 2b given in question of times t= 0s, 0.03s and 0.06s.

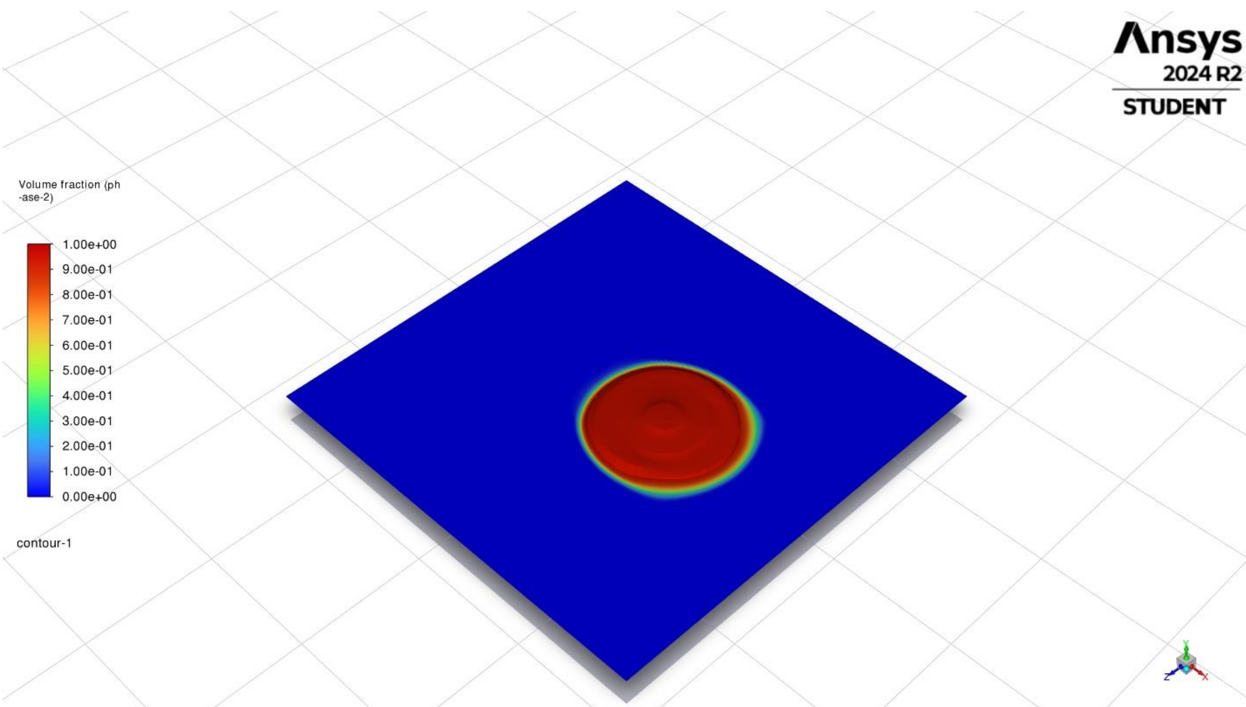
t= 0 s



t=0.03s

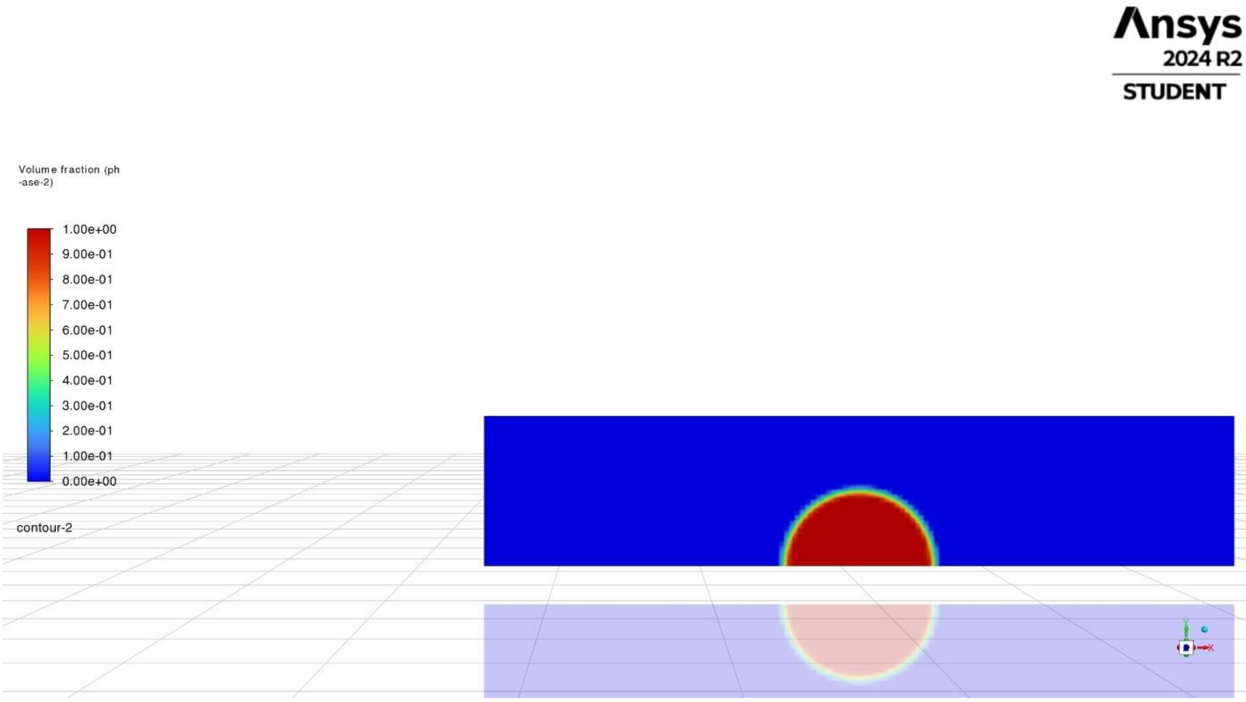

Ansys
2024 R2
STUDENT
Volume fraction (ph
-ase-2)
1.00e+00
9.00e-01
8.00e-01
7.00e-01
6.00e-01
5.00e-01
4.00e-01
3.00e-01
2.00e-01
1.00e-01
0.00e+00
contour-1

T=0.06s

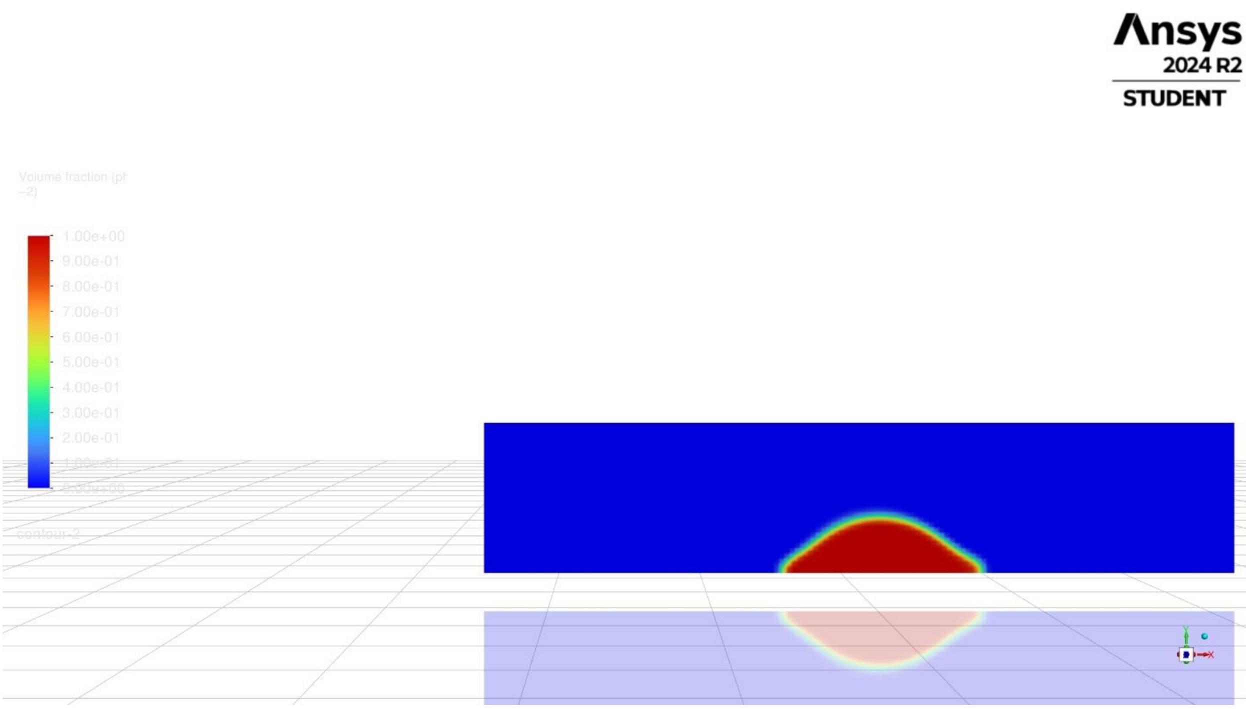

Ansys
2024 R2
STUDENT
Volume fraction (ph
-ase-2)
1.00e+00
9.00e-01
8.00e-01
7.00e-01
6.00e-01
5.00e-01
4.00e-01
3.00e-01
2.00e-01
1.00e-01
0.00e+00
contour-1

(D7) Three contour plots of volume fraction in plane of symmetry t= 0s, 0.03s and 0.06s.

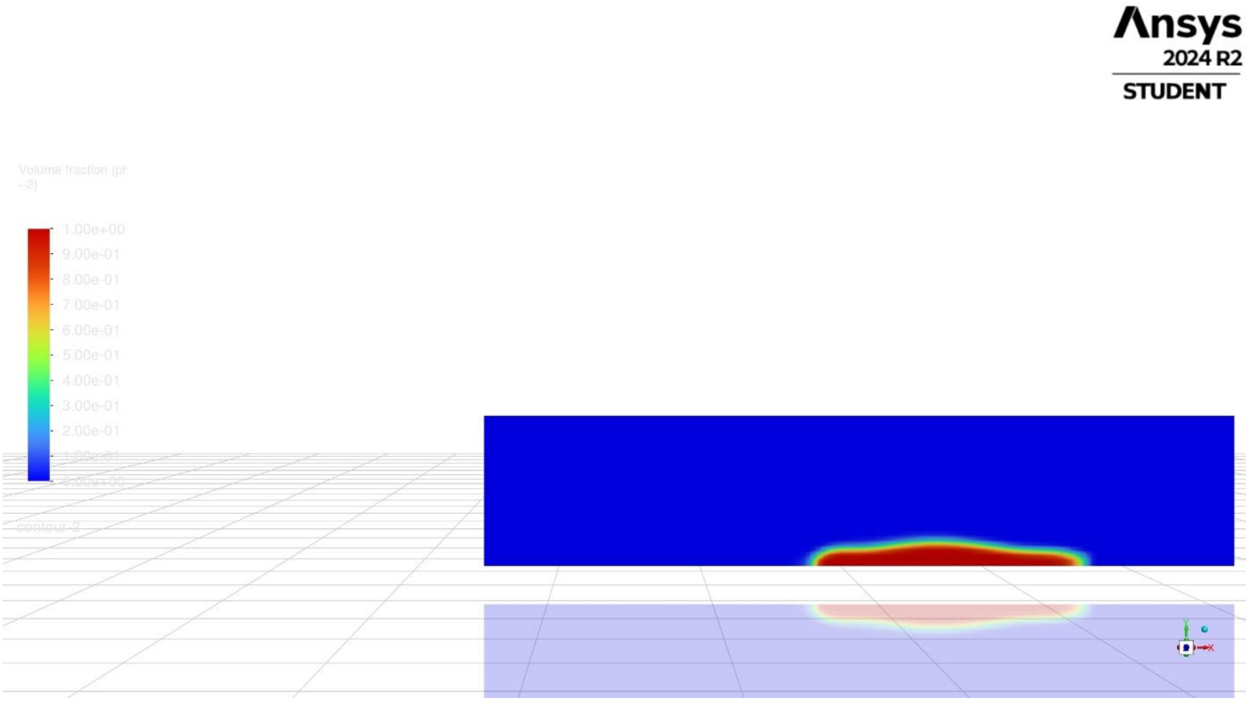
t=0 sec



t=0.03s



t=0.06s

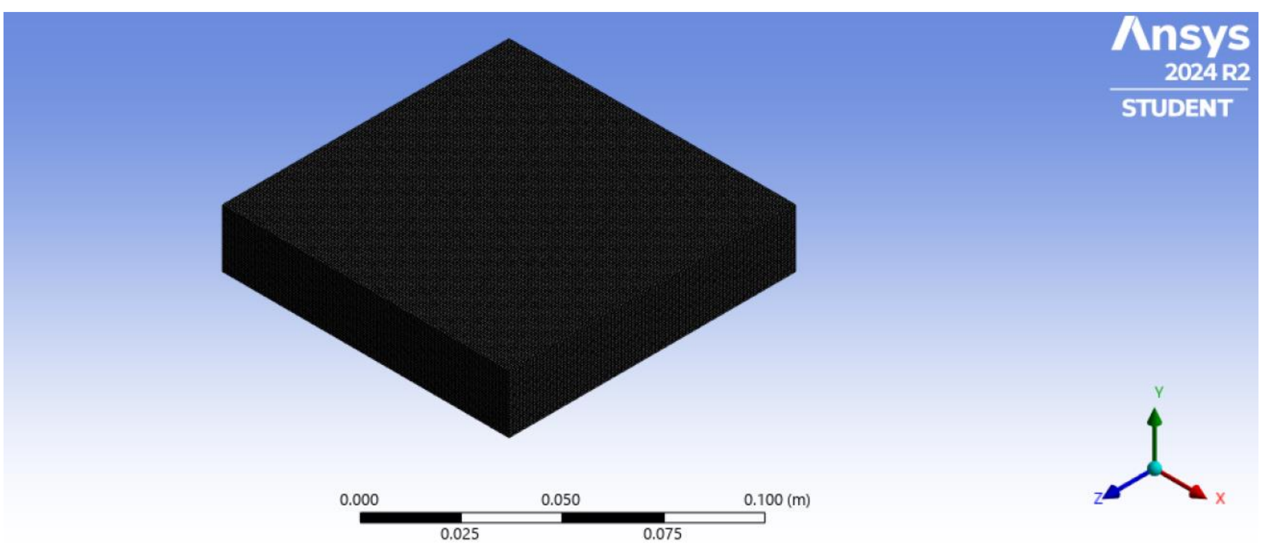


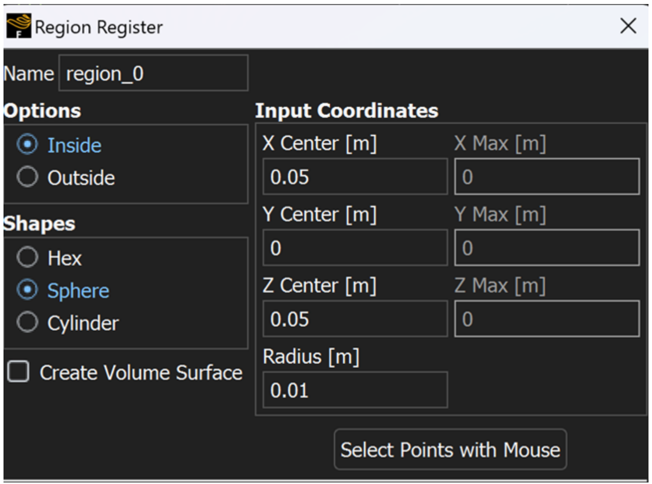
(D8)

Description of:

1. Computational Domain:

I have made a 10 cm* 10 cm box with depth of 2 cm.





Cell Register Used for droplet.

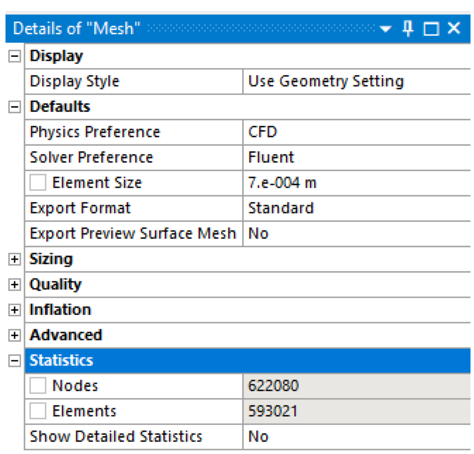
2. Boundary Conditions:

The boundary conditions used are:

For bottom plate, which is at 40 deg inclined angle, it is considered as 0 slip condition. For other walls this is filled with air.

3. Run Time

Number of time steps	Time step size	Max num of iterations
600	0.0001	20
300	0.0001	20
600	0.0001	20



Task 3:

1. A 3-D U-shaped pipe with air and water has two open top ends for free air movement.

2. Water oscillates in the pipe due to unequal initial levels, causing air flow at openings.

3. The oscillation period (τ) is monitored through mass flow rate, marking full cycles.

(a). Using gravity= -9.81 m/s^2

(D9).

The boundary conditions for two openings would be '**Pressure outlet**' with gauge pressure being '0' and no backflow.

Mesh Resolution:

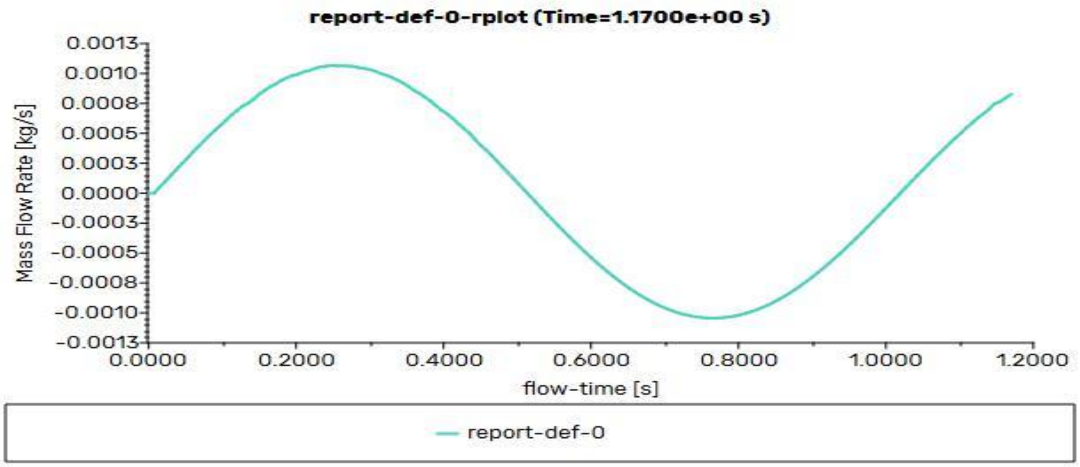
Element size used= 0.002m

Statistics	
Bodies	1
Active Bodies	1
Nodes	558245
Elements	540580
Mesh Metric	None

Run Parameters: for both Cases:

No of Time Steps	Time Step Size	Max Number of time steps
100	0.0075	10
100	0.0075	10
300	0.0075	10

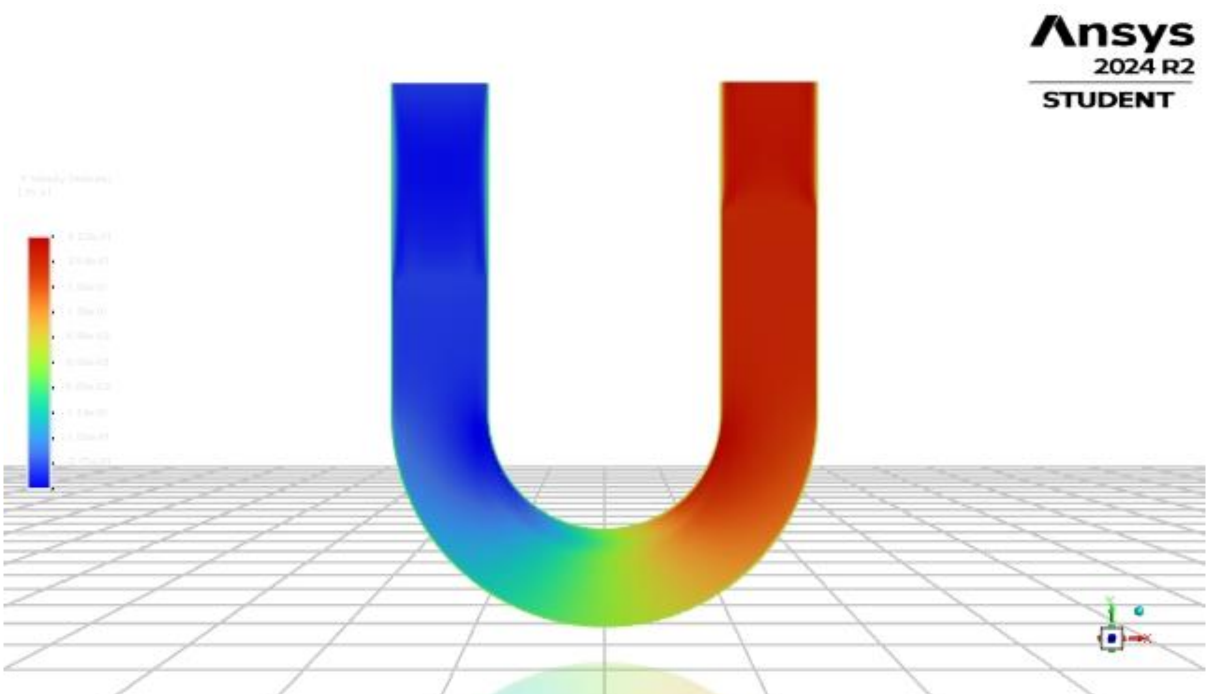
(D10). Plot of Mass flow rate of air in left opening



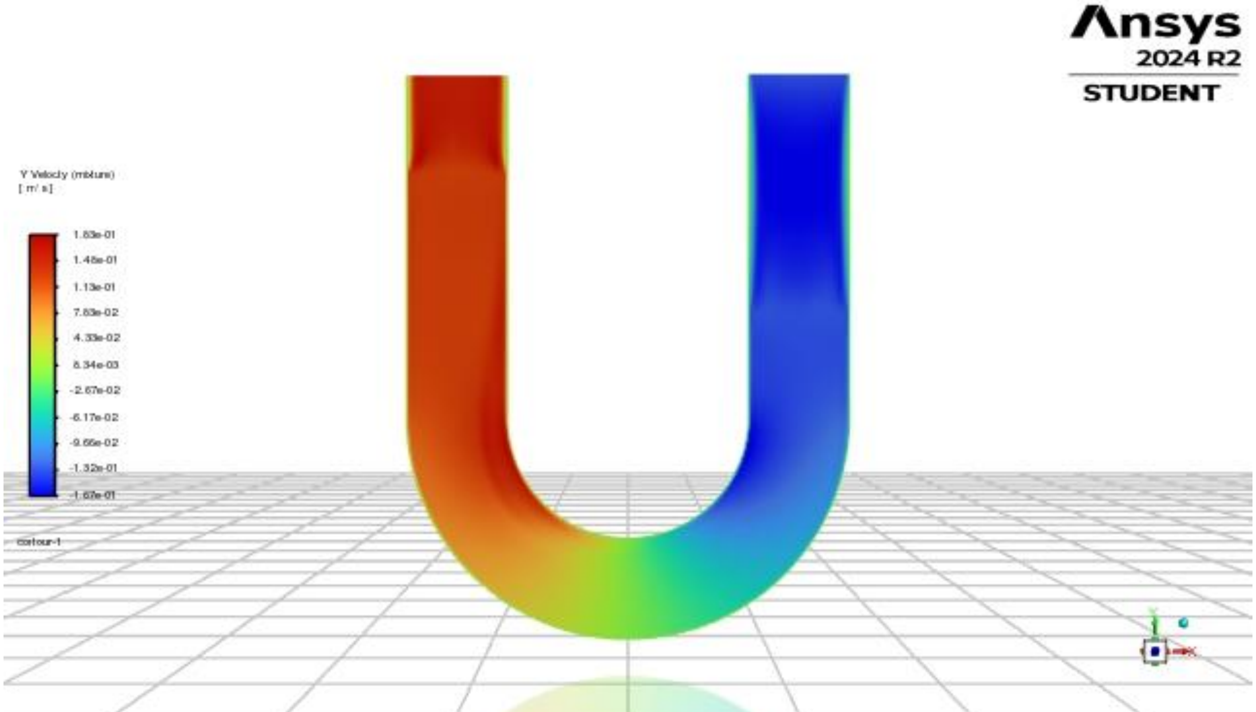
In the above plot the value of τ, which is length of one period of oscillation is **1.17** which can be rounded of to **1** as given in the question.

(D11). Contour plots of the y-velocity of the mixture on plane of symmetry at T=0.25 τ and 0.75 τ

1. At 0.25 τ

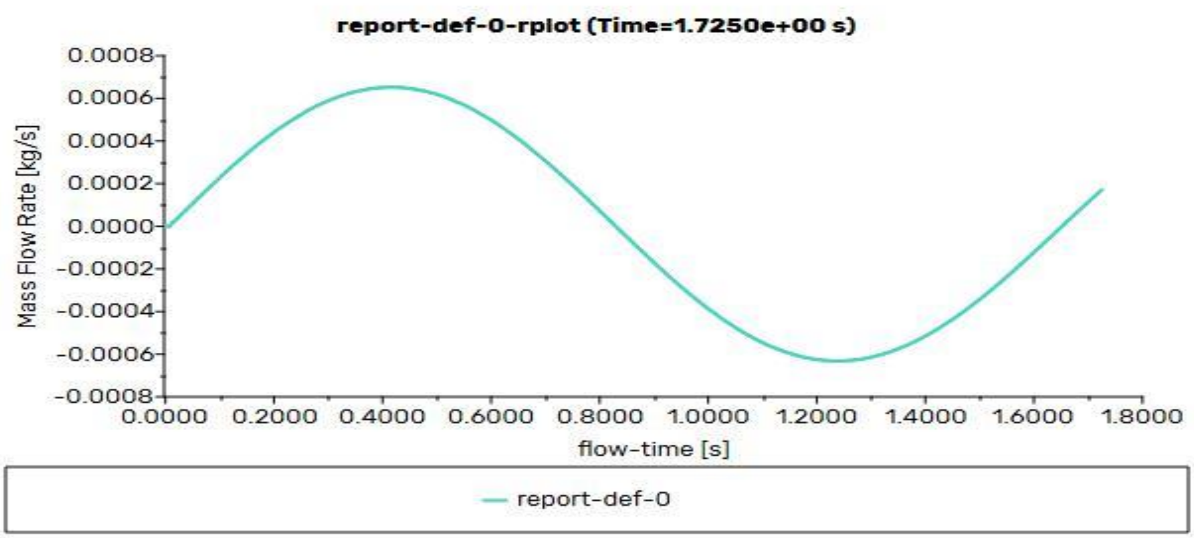


2. At t=0.75 τ



(b) Using gravity on mars = -3.72 m/s^2.

(D12). A plot of mass flow rate of air through left opening.



In the above plot the value of τ, which is length of one period of oscillation is 1.72 which can be rounded of to 1 as given in the question. The oscillation period is extended on Mars compared to Earth, increasing from 1 second to 1.7 seconds, primarily due to differences in gravitational strength. Gravity plays a crucial role in fluid dynamics, especially when buoyant forces and other gravity-dependent effects are at play, such as in your U-shaped pipe simulation. With Mars' weaker gravitational pull, buoyant forces acting on the air within the pipe are reduced. This diminished buoyancy results in slower acceleration and deceleration of the air as it moves through the pipe's curves and openings, prolonging the time needed to complete a full oscillation cycle.

Task4

We simulate the process of a falling engine oil droplet colliding with a flat surface of water, in the 2-D setting. The geometry of the system is a simple 25 cm x 25 cm square bucket that is open (to air) at the top and with the other three sides being walls.

Considering g=-9.81 m/s^2, Setting surface tension coefficient to *0.072 N/m* for the water-air interface*, 0.03 N/m* for the interface between engine oil and air, and *0.03 N/m* for the interface between engine oil and water.

I have used, custom field function =

*CF = 0.1*VF1 + 0.2*VF2 + 0.3* VF3*

(D13)

Mesh Resolution:

Statistics	
Bodies	1
Active Bodies	1
Nodes	19177
Elements	18900
Mesh Metric	None

Element Size: 0.002m

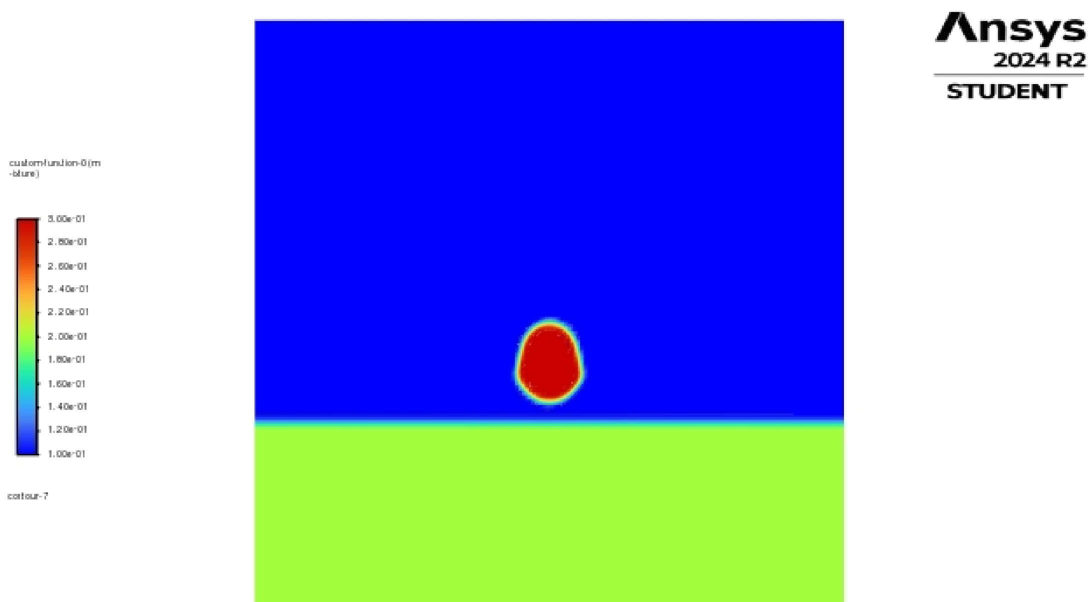
Run Parameters:

Time	Time step size	Max number of iterations	Number of time steps
0.14s	**0.001**	**20**	**140**
0.18s	**0.001**	**20**	**180**
0.22s	**0.001**	**20**	**220**

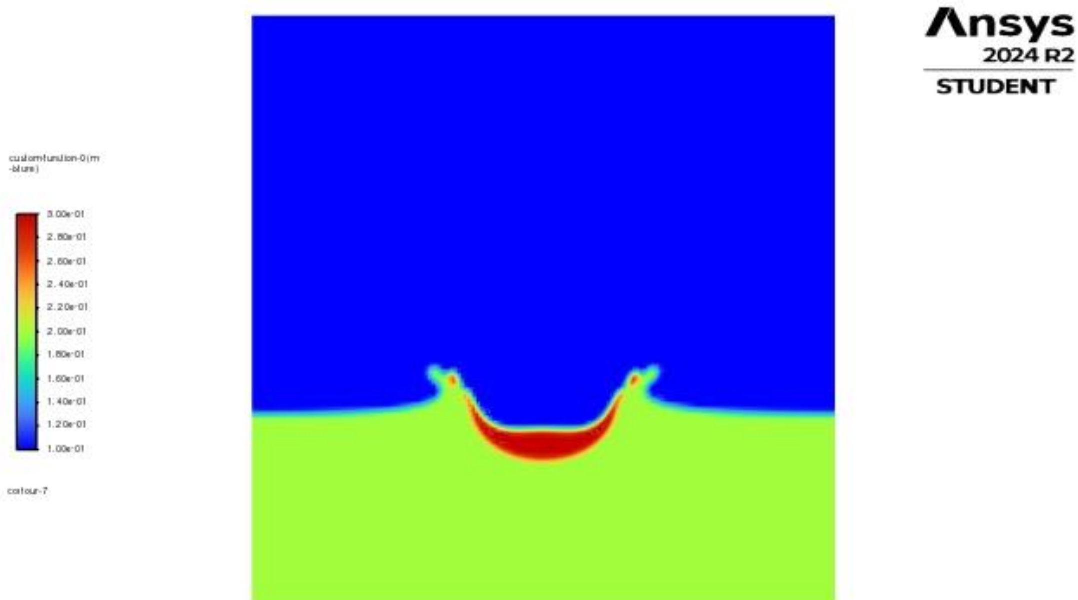
(D14)

Contour plots of the *custom field function* CF at t= 0.14s, 0.18s and 0.22s.

At t = 0.14s



At t = 0.18s



At t = 0.22s