

MAE 560 Applied CFD

Project 2

Statement of collaboration

This statement affirms that I didn't collaborate with anyone for this project

Barath Sundaravadivelan

Task-1

(D1) (Case A) Contour plots of the volume fraction of methane at $t = 3$ s and $t = 6$ s.

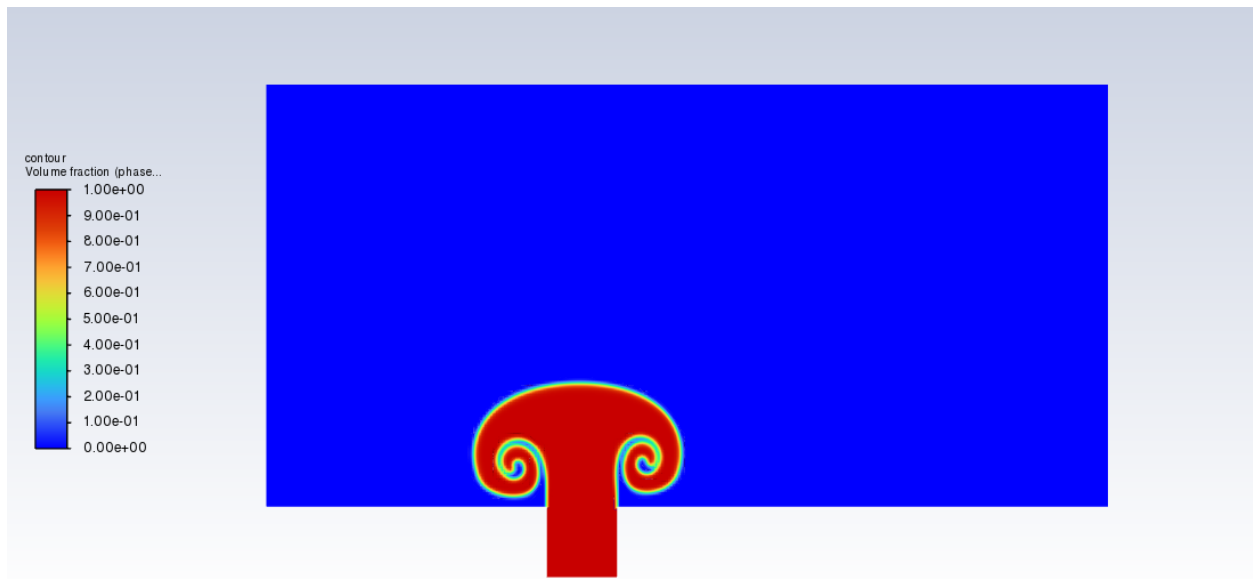


Figure 1 VOF of Methane at $t = 3$ sec

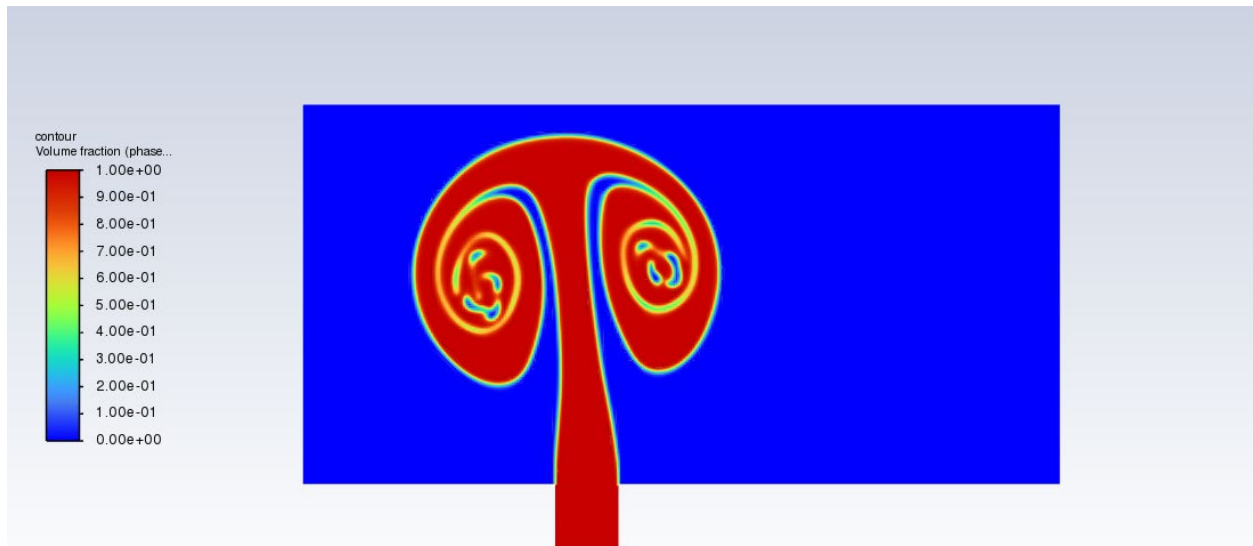


Figure 2 VOF of Methane at $t = 6$ sec

(D2) (Case B) Contour plot of the volume fraction of methane at $t = 6$ s.

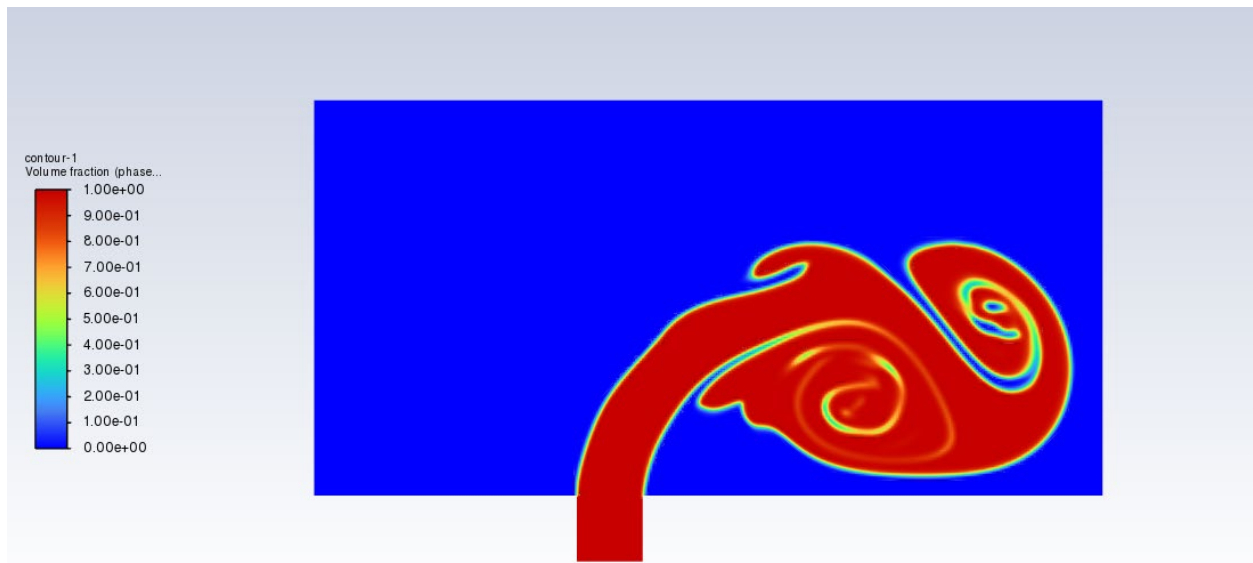


Figure 3 VOF of Methane at $t = 6$ sec

(D3) Descriptions: Same for both Case A and Case B

- **Mesh resolution:** 20 cm
- **Time step size:** 0.01 sec
- **Maximum number of iterations per time step:** 10

Task-2

(D4) Descriptions:

- **Mesh resolution:** 0.05 cm
- **Time step size:** 0.0004 sec
- **Maximum number of iterations per time step:** 10

(D5) Contour plots of custom field function CF:

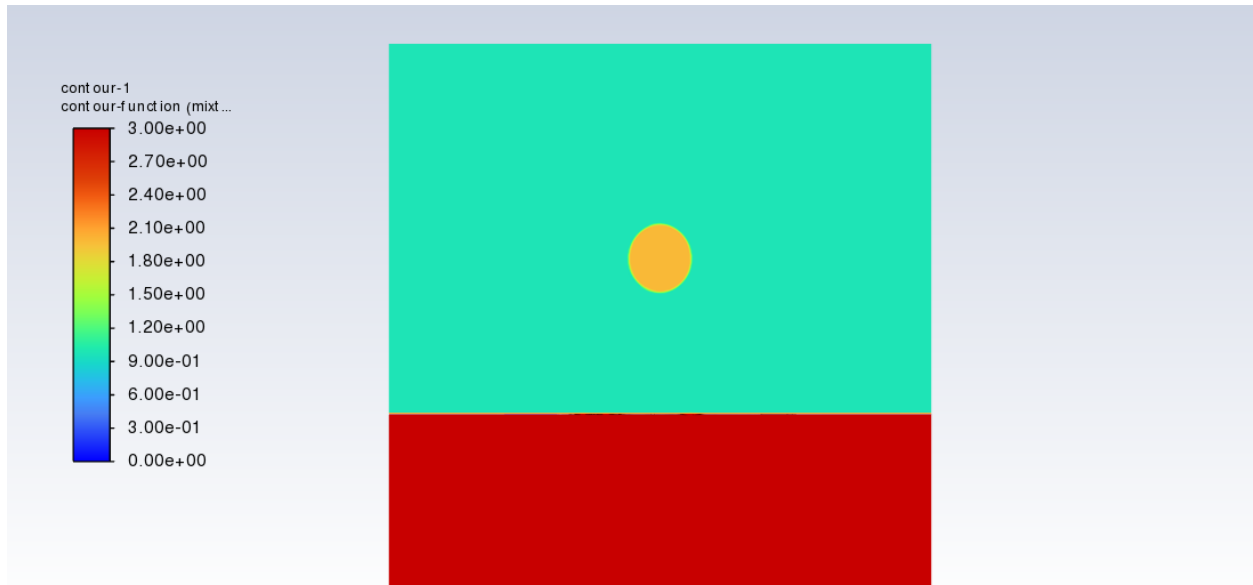


Figure 4 Contour plot of custom field VOF at $t = 0.1$ sec

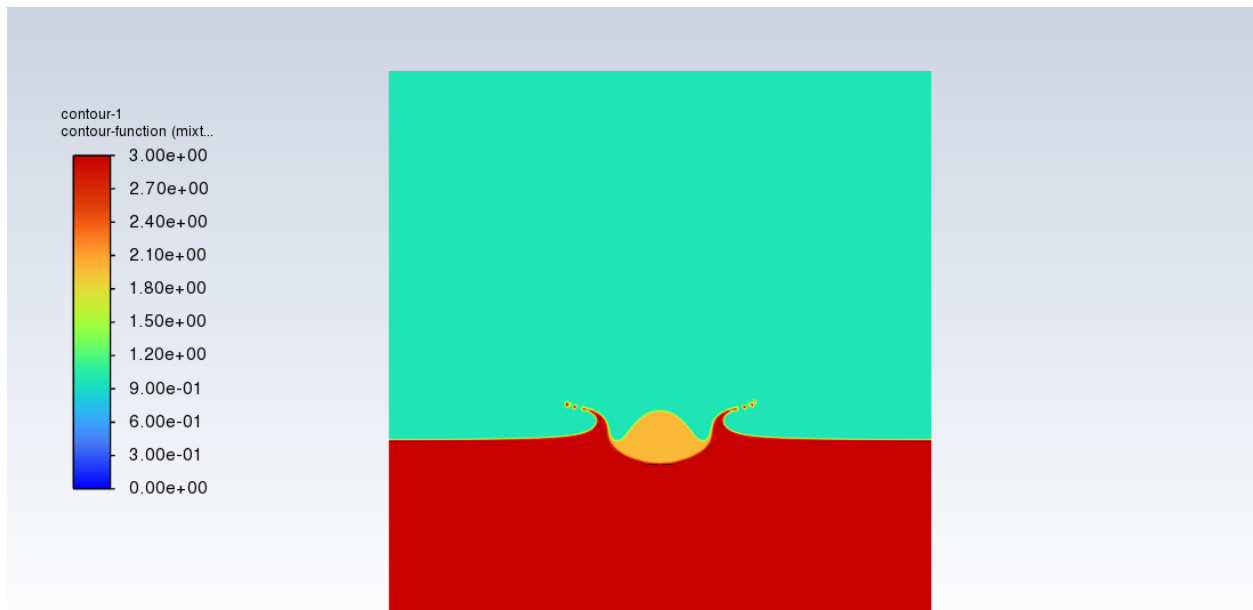


Figure 5 Contour plot of custom field VOF at $t = 0.16$ sec

contour-1
contour-function (mixt...
3.00e+00
2.70e+00
2.40e+00
2.10e+00
1.80e+00
1.50e+00
1.20e+00
9.00e-01
6.00e-01
3.00e-01
0.00e+00

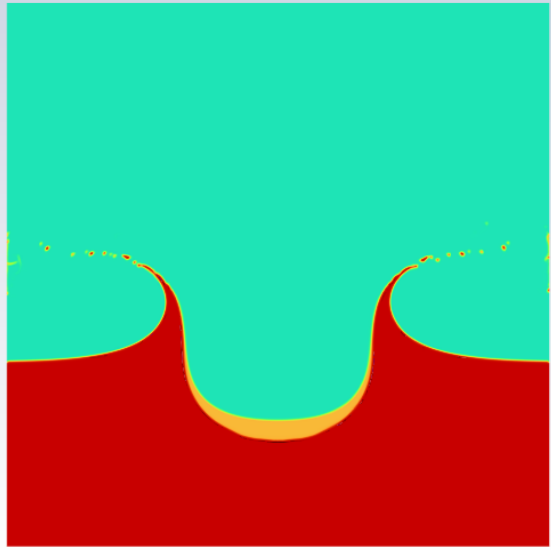


Figure 6 Contour plot of custom field VOF at t= 0.2 sec

Task-3

(D6) Descriptions:

- **Computational domain:** It is a 20x10x5 cm cuboid. To conserve the 500K element limit in the student version of Ansys, we try to keep the size of computational domain as small as possible. In order to improve the accuracy of the result, we predict the expected path in which the glycerin would glide and refine the mesh along that path.
- **Boundary conditions:** Bottom side of the domain is set to be wall, all the other 5 sides are set to pressure outlet with zero pascal to emulate open air condition.
- **Mesh resolution (pre-adapt):**
Element size: 0.25cm
Max size: 0.5cm
- **Time step size:** 0.0005 sec
- **Maximum number of iterations per time step:** 10

(D7) Isometric view of the iso-surface and the floor at $t = 0, 0.08$ s, and 0.2 s.

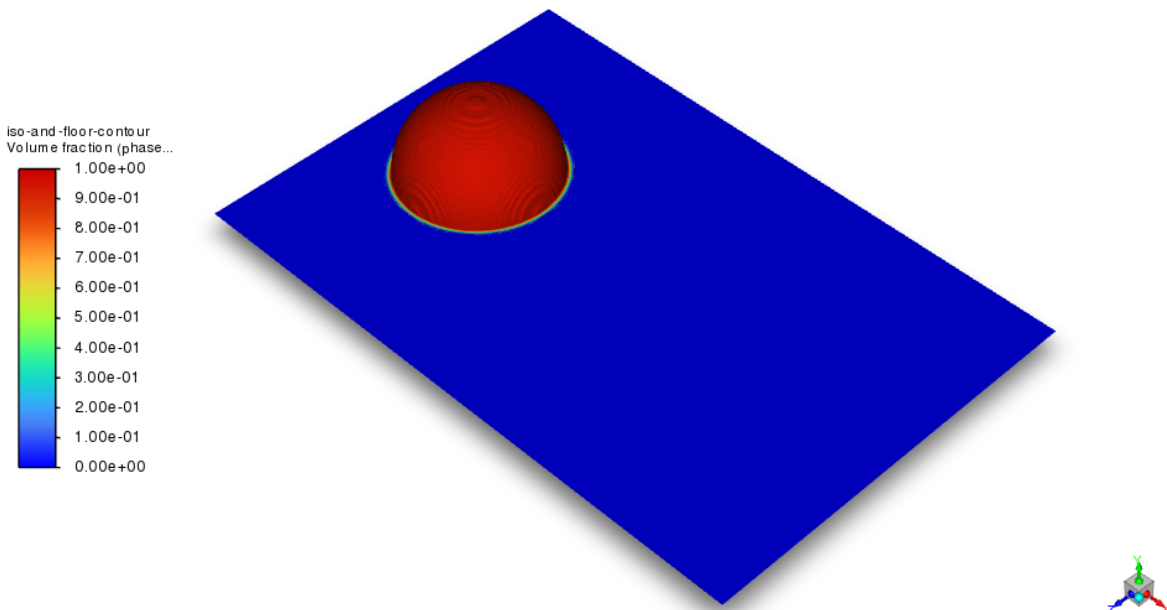


Figure 7 Contour of the iso-surface at $t = 0$ sec

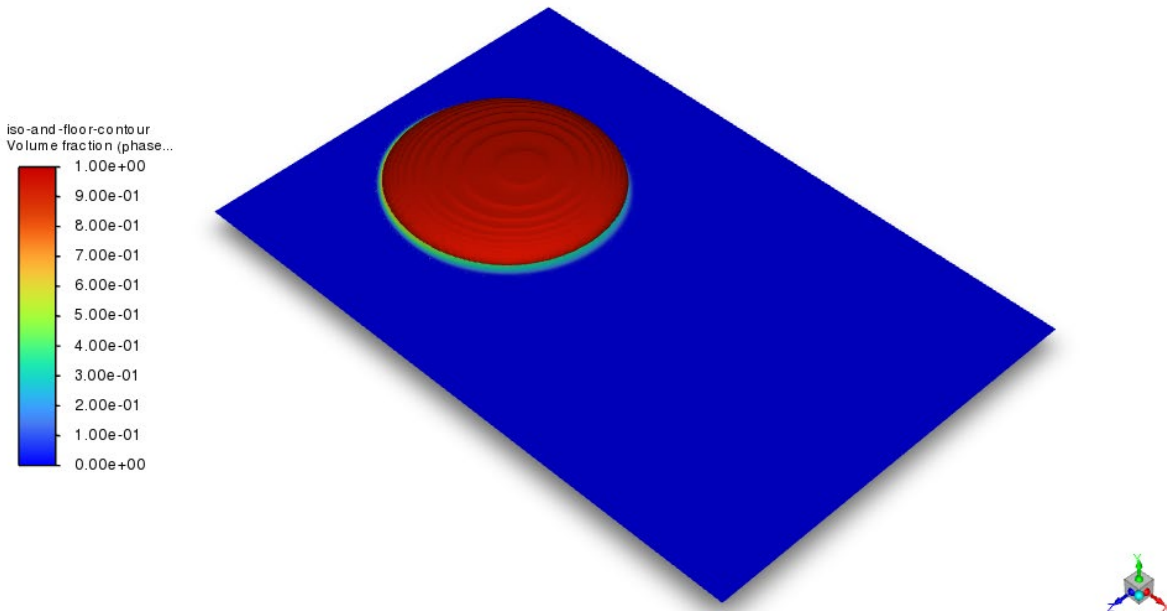


Figure 8 Contour of the iso-surface at $t= 0.08$ sec

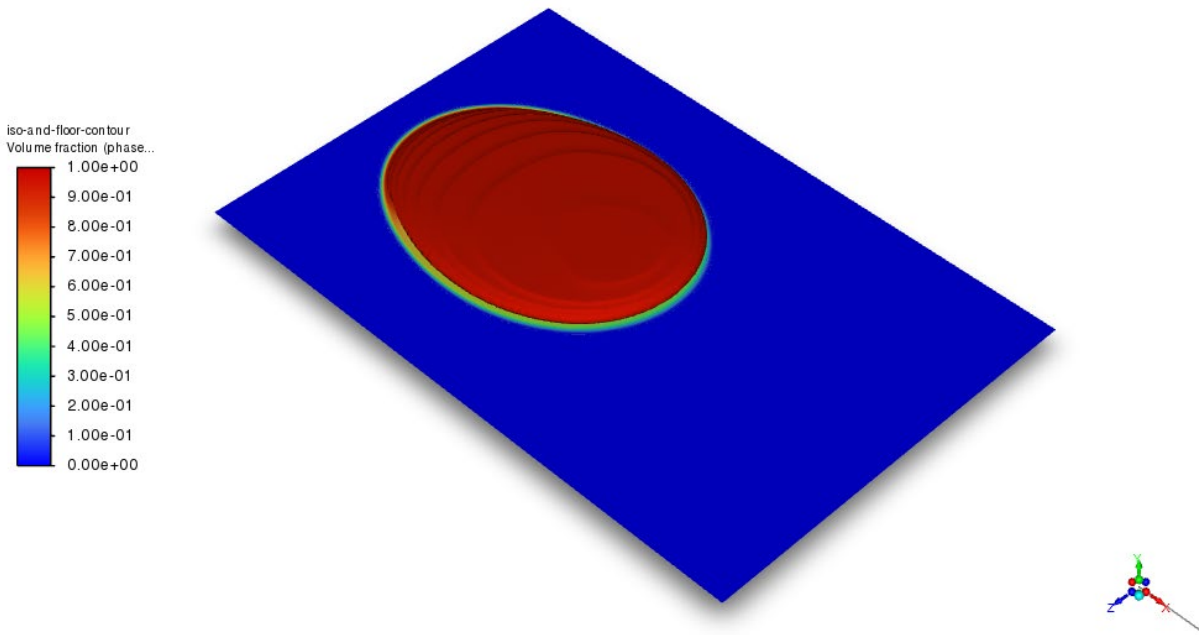


Figure 9 Contour of the iso-surface at $t= 0.2$ sec

(D8) Contour plots of VOF of glycerin on the plane of symmetry at $t = 0, 0.08$ s, and 0.2 s.

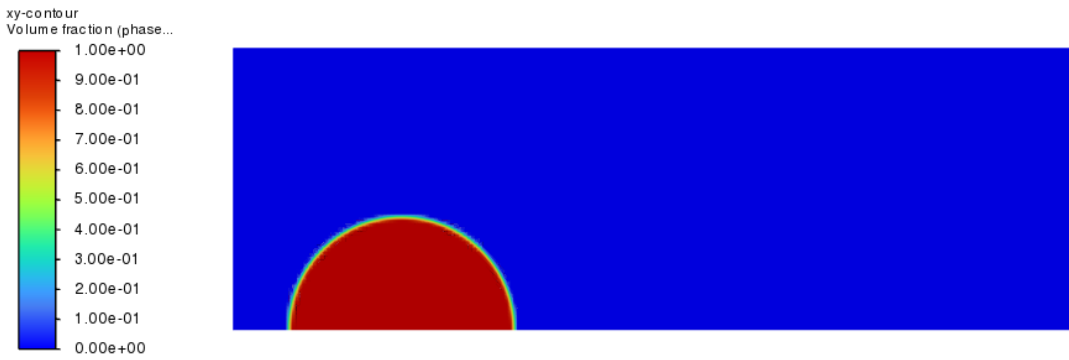


Figure 10 Contour of VOF of glycerin at $t = 0$ sec.

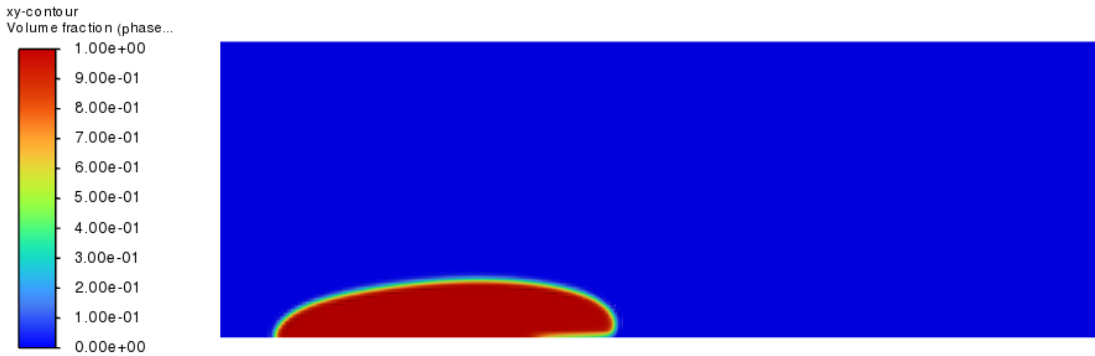


Figure 11 Contour of VOF of glycerin at $t = 0.08$ sec.

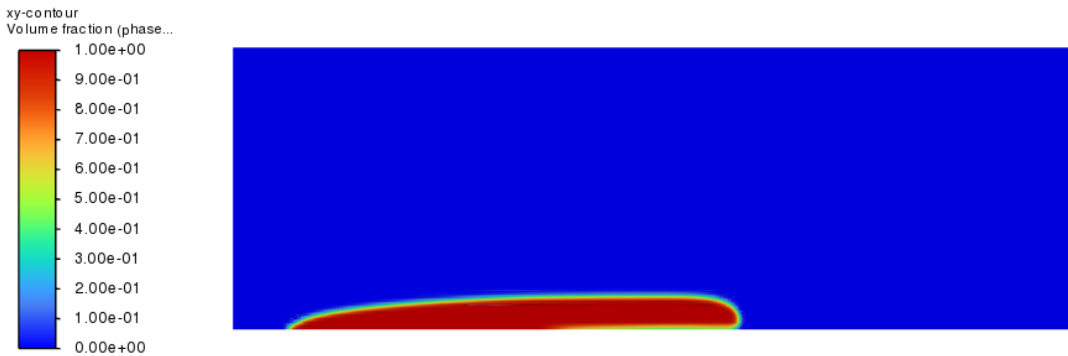


Figure 12 Contour of VOF of glycerin at $t = 0.2$ sec.

Task-4

(D9) Descriptions:

- **Boundary conditions:** Both top and bottom side are set to pressure outlet with zero pascal to emulate open to air condition.
- **Mesh resolution (pre-adapt):** Element size: 0.8cm
Max size: 0.9cm
- **Time step size:** 0.05 sec
- **Maximum number of iterations per time step:** 10

(D10) Time for half the volume to drain out:

	Initial VOF	VOF nearest to 50%	Percentage drained	Time taken (sec)
Water	0.0230911	0.01153311	50.05387357	$t_A = 7.25$
Engine oil	0.0230911	0.01154001	50.02399193	$t_B = 11.55$

Table 1: Data gained from time step= 0.05 sec

Note: In order to have a 0.1 accuracy, time step of 0.05 is considered to ensure one step higher order of accuracy. This is discussed a bit more at the end.*

Water drains *faster* than Engine oil. This makes sense because water viscosity is lower than engine oil.

(D11) Vector plot of velocity magnitude at a plane 15cm from the top of the basin when half of the engine oil is drained:

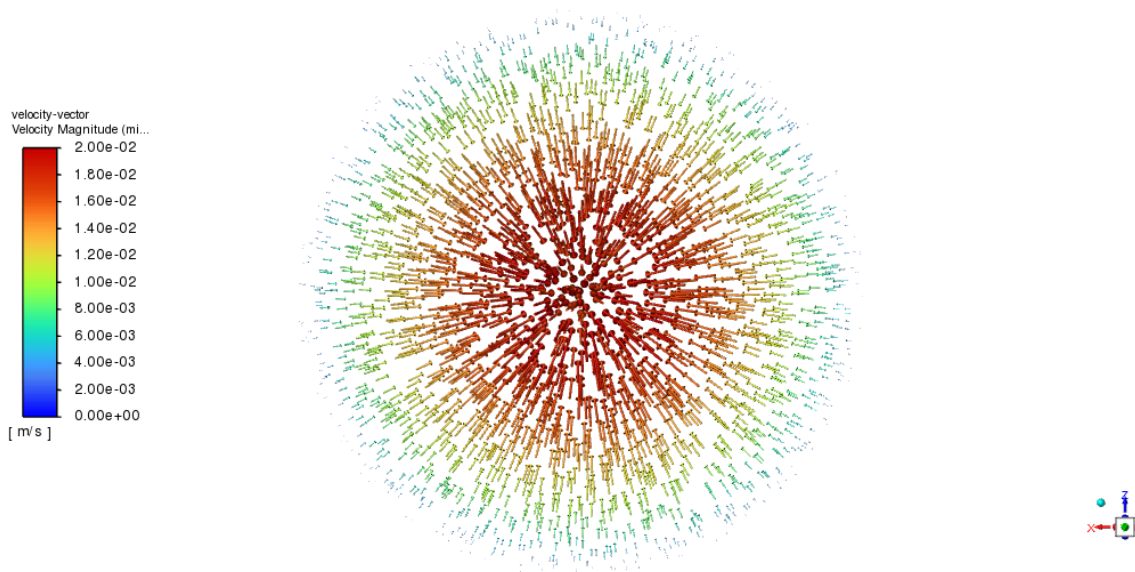


Figure 13 Vector plot of velocity magnitude at t=11.55 sec.

***Note: How mesh refinement and time-step considered changed the results**

Using Ansys student version limits the number of elements that can be computed to 500K. Which is more than enough for most of the project. Initially, I used a time step size of 0.1 sec since it is the order of accuracy expected. With all the parameters unchanged from the results shown above, I got $t_A = 8.1$ sec with 50.4303% of water drained. After acquiring the above data from with time step 0.05 sec, I was curious about how a refined mesh would affect the result. So, I ran another test with a refined mesh which had about 460K elements instead of 414K elements but yielded around the same 8.0 sec with 50.55021% water drained.

From this, I observed that the time step had a major impact on the result than the refinement of the mesh. Since 0.05 sec timestep yielded a close to 50% VOF result, I consider it my final solution.

But just for good measure, I am uploading all the data corresponding to (D10) and (D11) that I got from using the 0.1 sec time step.

	Percentage drained	Time taken (sec)
Water	50.4303	$t_A = 8.1$
Engine oil	50.2928	$t_B = 12.2$

Table 2: Data gained from time step= 0.1 sec

Again, water drained faster than engine oil.

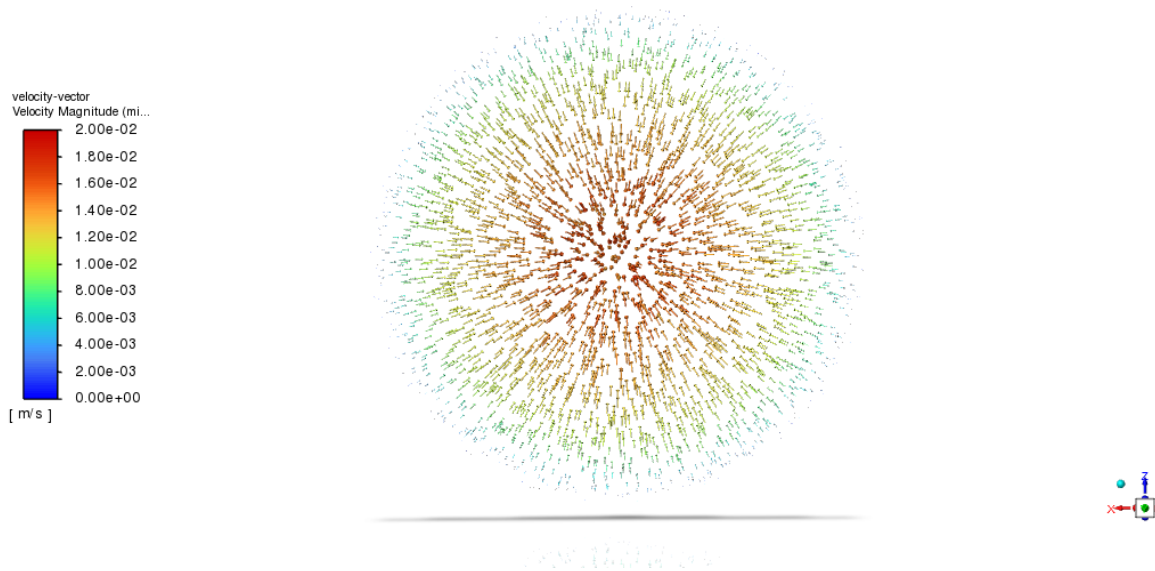


Figure 14 Vector plot of velocity magnitude at $t=12.2$ sec.