Project 3

MAE 598: Applied Computational Fluid Dynamics

Molly Rhodes

Name of collaborator: Adam Paredes	
Tasks	Contribution
Task 1	Worked on geometry, mesh, setup, and discussed results
Task 2	Worked on geometry, mesh, setup, and discussed results
Task 3 (a&b)	Worked on geometry, mesh, setup, and discussed results
Task 4	No collaboration

Task 1:

To simulate the water droplet falling into water, the square tank geometry was created and a mesh with linear element order and an element size of 0.005 m was generated. An implicit scheme was selected in the volume fraction parameters. The top boundary was set as a pressure outlet with zero gauge pressure and the backflow phase was set to air. The surface tension coefficient was taken from the Fluent database, and it was equal to 0.0719404 N/m. The water in the tank and the droplet were created using the patch option after initializing the solution. The solution ran for 500 time steps with a time step size of 0.001 seconds and the maximum number of iterations per time step was 20. The contours in figures 1-4 show how the droplet deforms as it falls and how the water in the tank deforms when the droplet makes contact with the water.

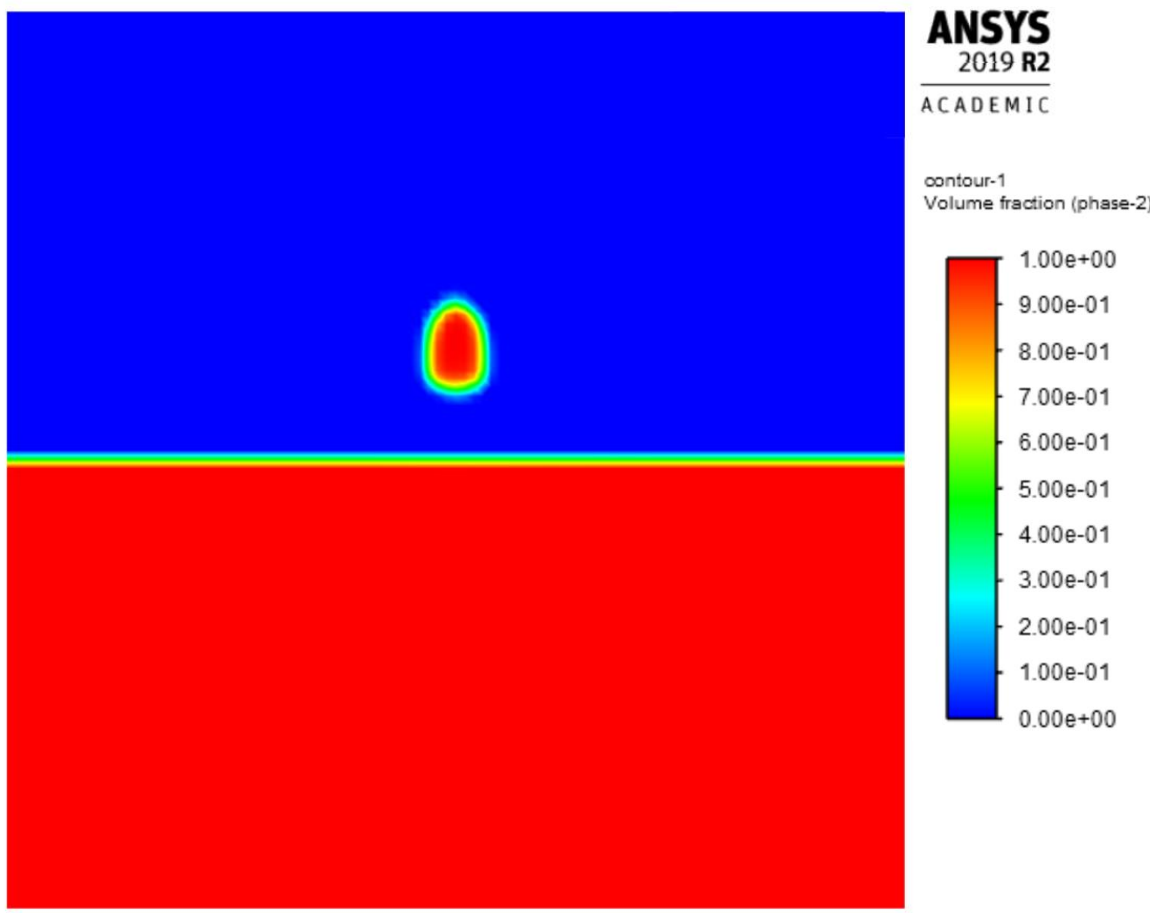

Figure 1: Contour plot of the volume fraction of water at 0.15 s

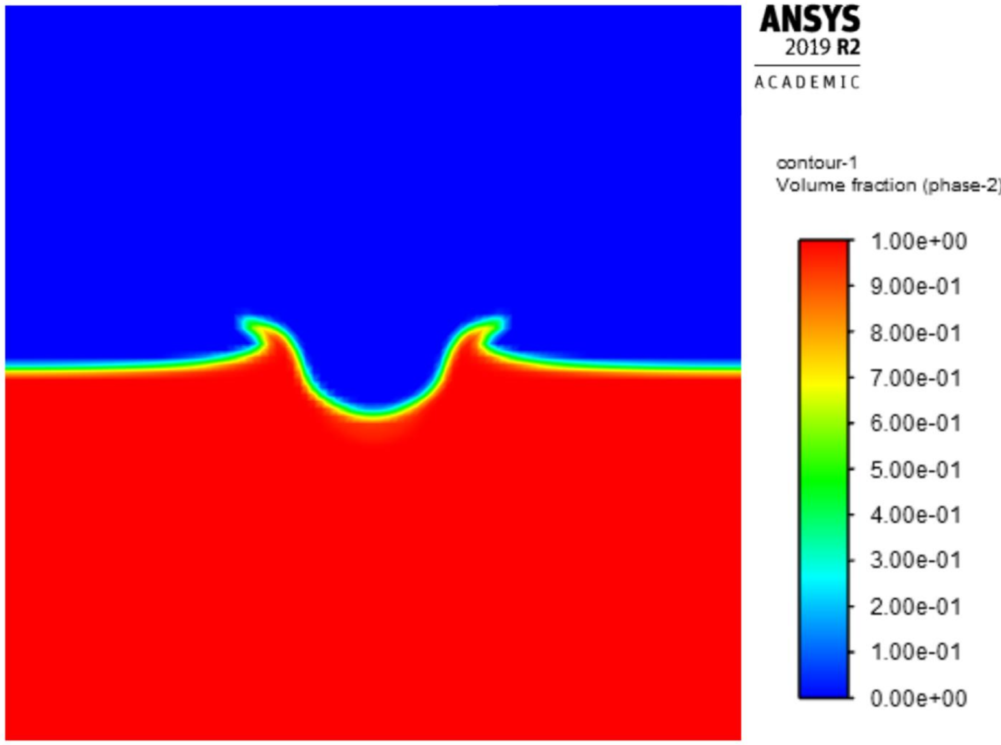

Figure 2: Contour plot of the volume fraction of water at 0.25

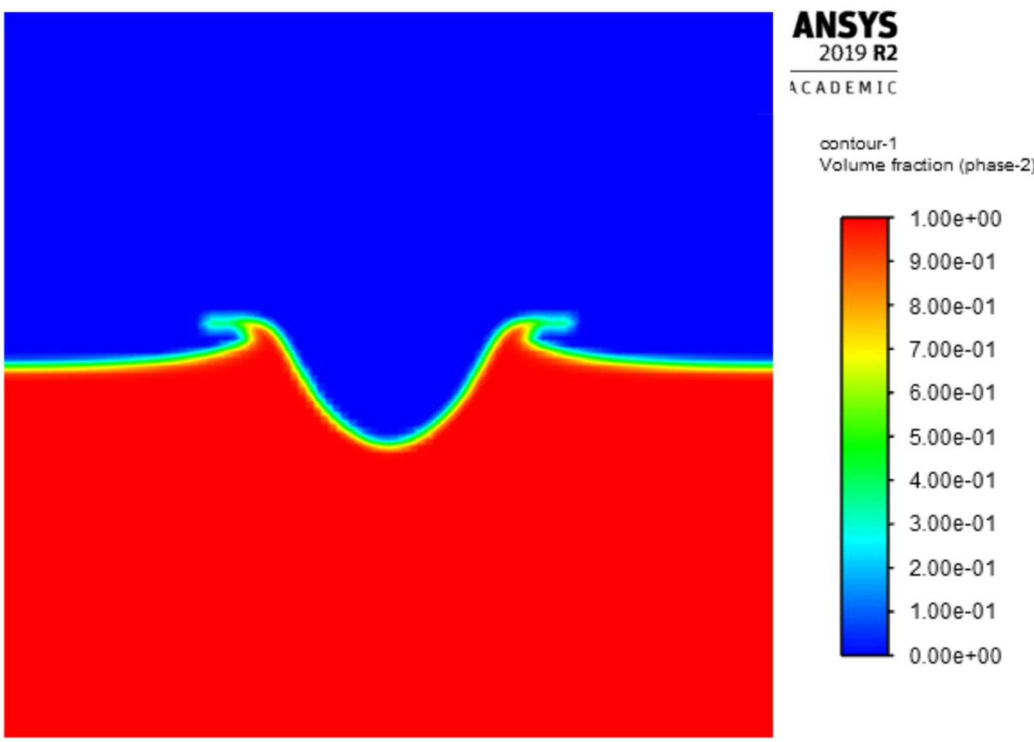

Figure 3: Contour plot of the volume fraction of water at 0.3 s

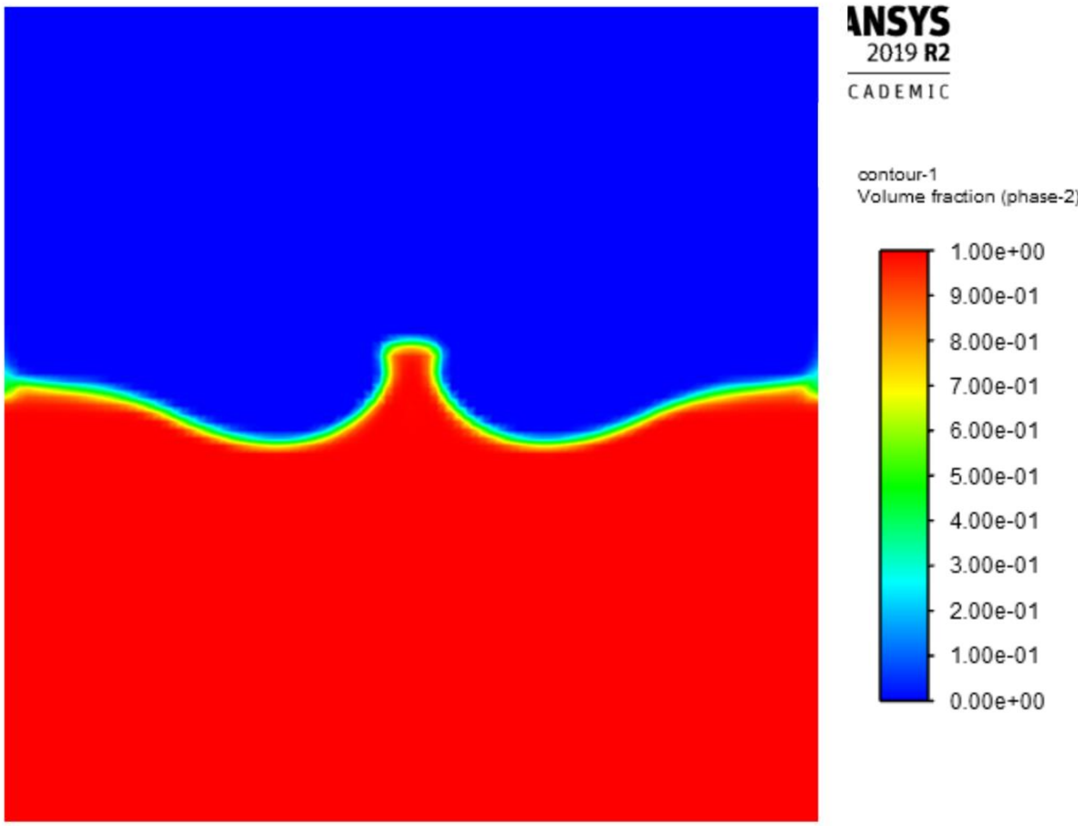

Figure 4: Contour plot of the volume fraction of water at 0.5 s

Task 2

The geometry for this task was created with a box at an angle of 40° from the ground. The dimensions of the plate were 12.5 cm x 12.5 cm, and it was extruded 3 cm in the normal direction. The mesh used for this simulation had elements with a linear order and a size of 0.002 m. The boundary conditions on the walls and top of the box were prescribed as pressure outlets with zero gauge pressure, and the bottom surface of the box was defined as a wall. An implicit method was selected in the volume of fluid options. The time step size used was 0.001 seconds for 100 time steps with a maximum of 20 iterations per time step. As the time increases, it can clearly be seen that gravity causes the glycerin blob to flatten and move down the ramp.

i.

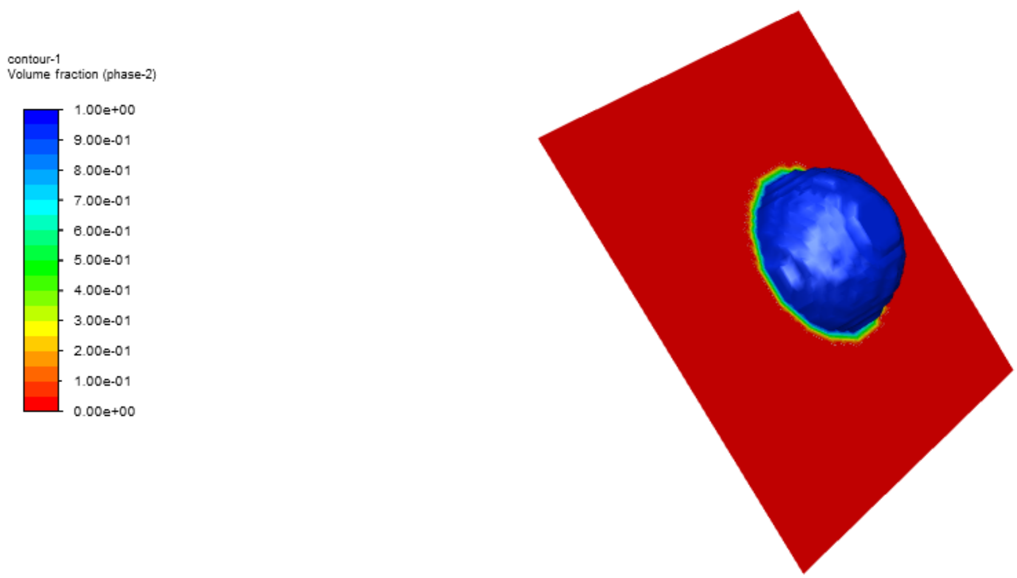

Figure 5: Contour plot of the volume fraction of glycerin at 0 s

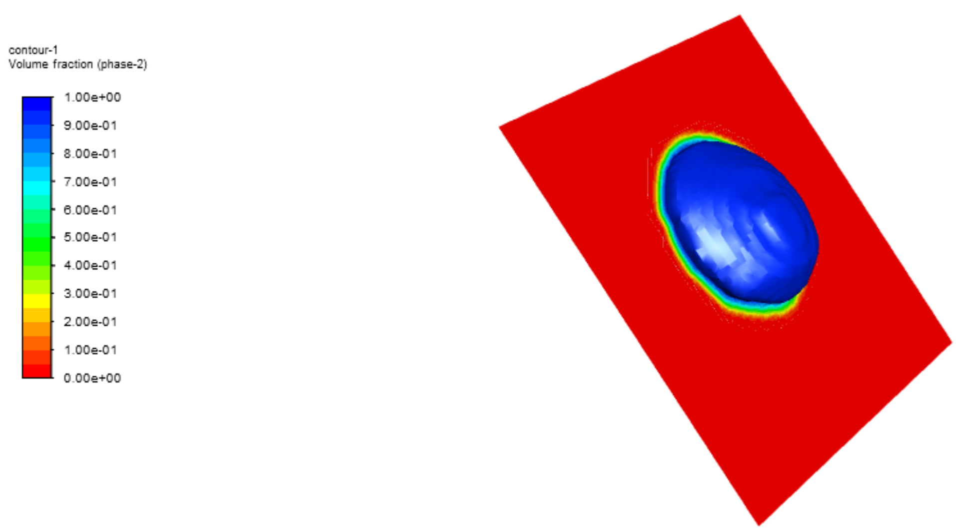

Figure 6: Contour plot of the volume fraction of glycerin at 0.05 s

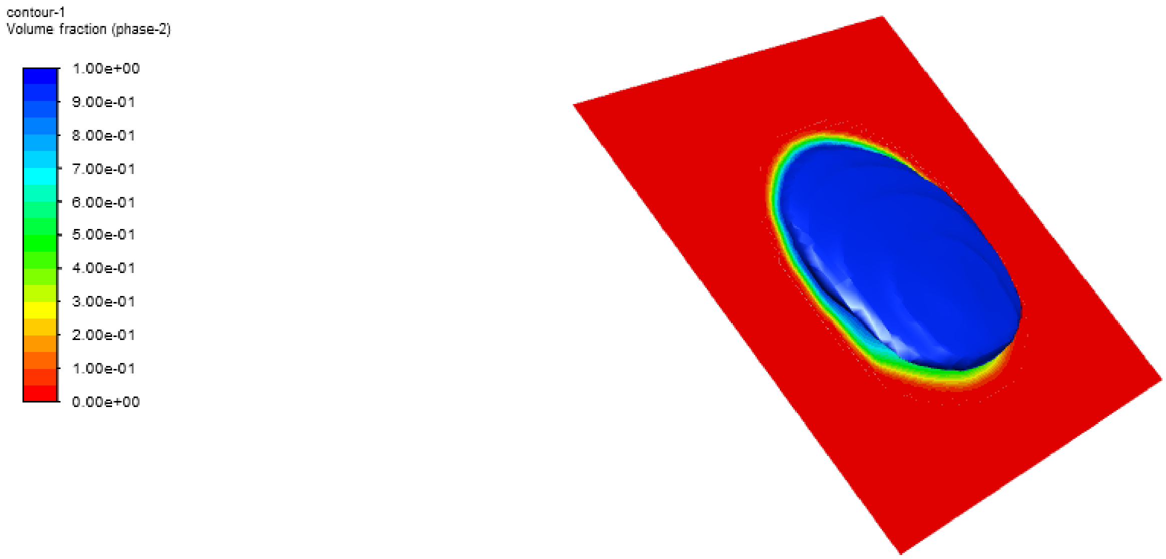

Figure 7: Contour plot of the volume fraction of glycerin at 0.1 s

ii.

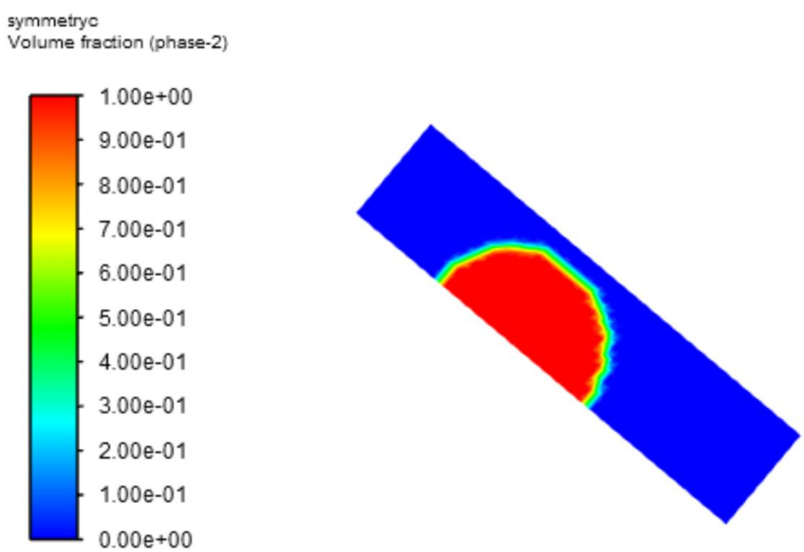

Figure 8: Contour plot of the volume fraction of glycerin on the plane of symmetry at 0 s

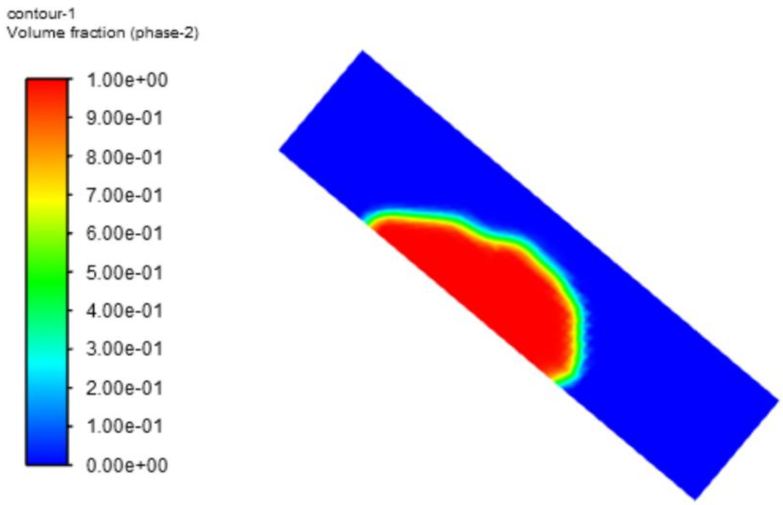

Figure 9: Contour plot of the volume fraction of glycerin on the plane of symmetry at 0.05 s

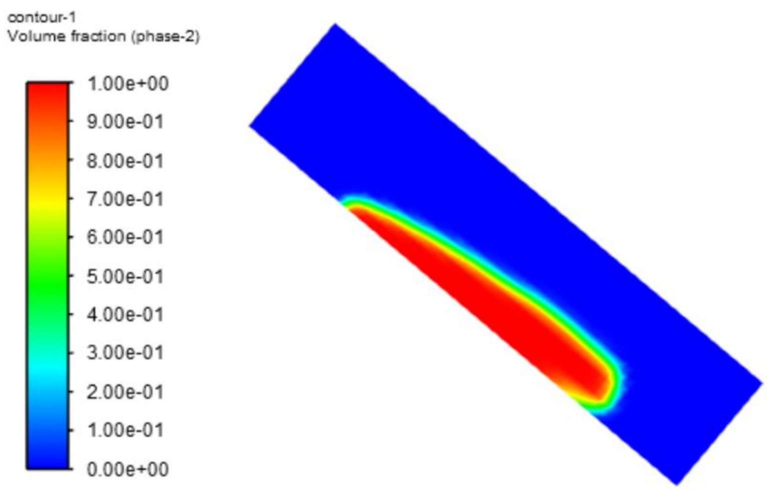

Figure 10: Contour plot of the volume fraction of glycerin on the plane of symmetry at 0.1 s

Task 3a:

To simulate the leaking of methane in open air the geometry described in the problem statement was created, and a mesh with quadratic order elements with a size of 0.35 m was generated. An implicit scheme was used within the volume of fluid method. For this task, the three side boundaries were all set as pressure outlets with no gauge pressure and the backflow phase was set to air. The entrance of the side pipe was set as a pressure inlet with a gauge pressure of 50 Pa, and the methane was pumped into the system through this inlet. The simulation ran for 1000 time steps with a time step size of 0.01 s. Initially the system was filled with air, and the methane entered the system as time increased.

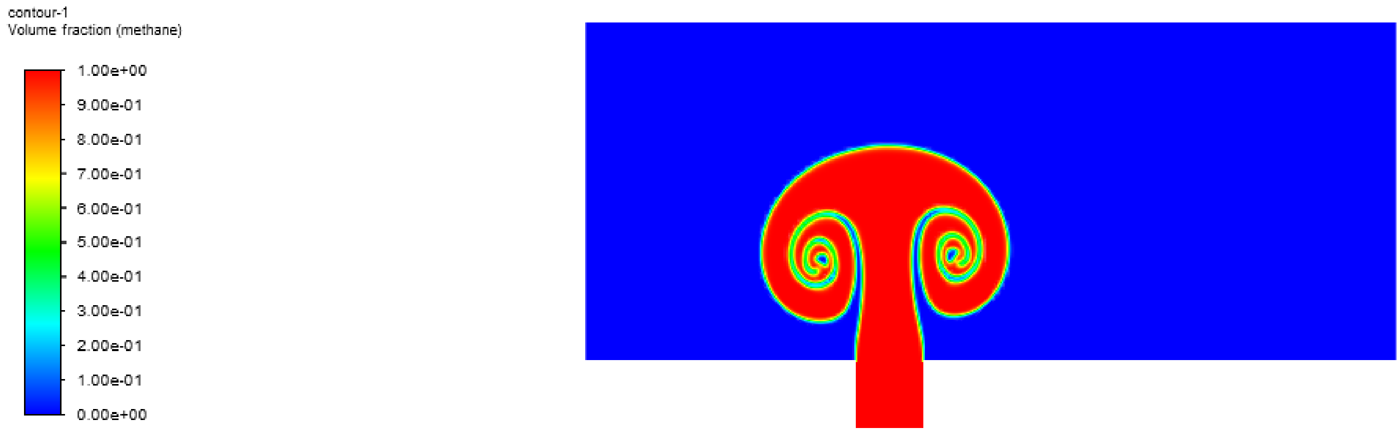

Figure 11: Contour plot of the volume fraction of methane at 7 s

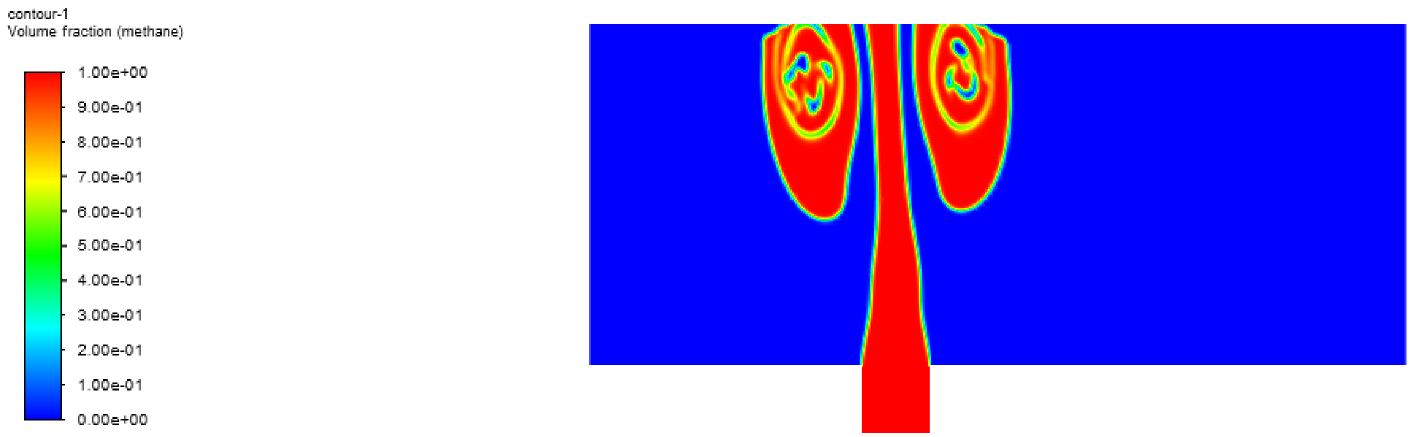

Figure 12: Contour plot of the volume fraction of methane at 10 s

Task 3b:

The same general setup from task 3a was used for this task as well. The only change is that the left boundary was changed from a pressure outlet to a velocity inlet with a velocity profile of $u = 0.2y$ for the x-velocity. This caused the methane cloud to be blown to the right as it moved into the system.

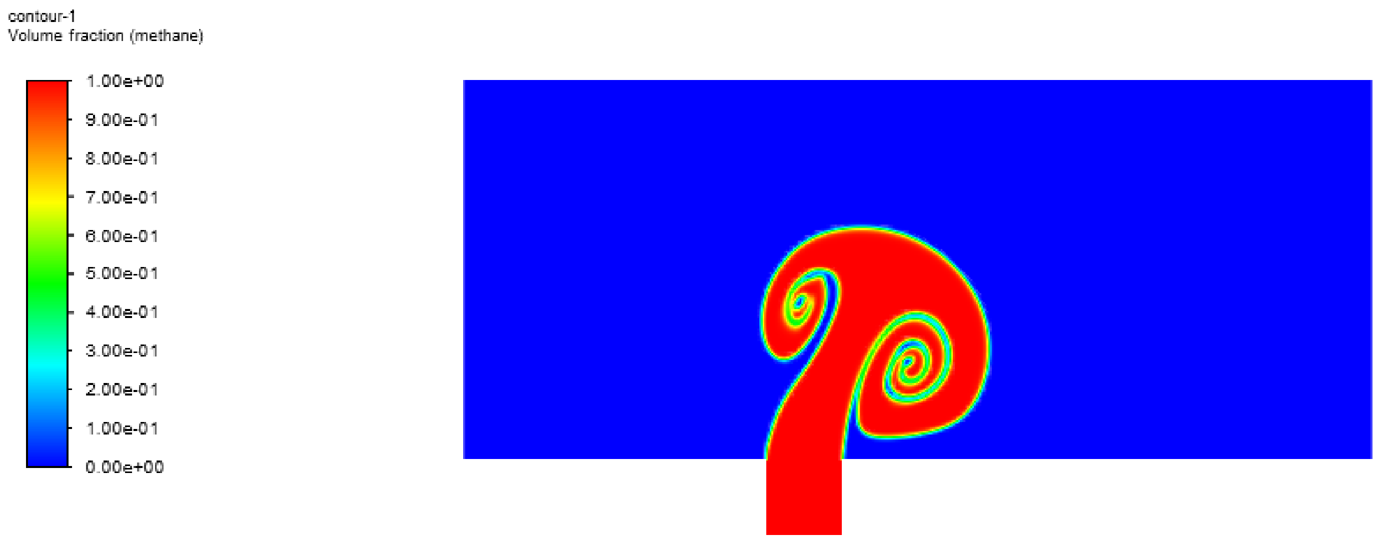

Figure 13: Contour plot of the volume fraction of methane at 9 s

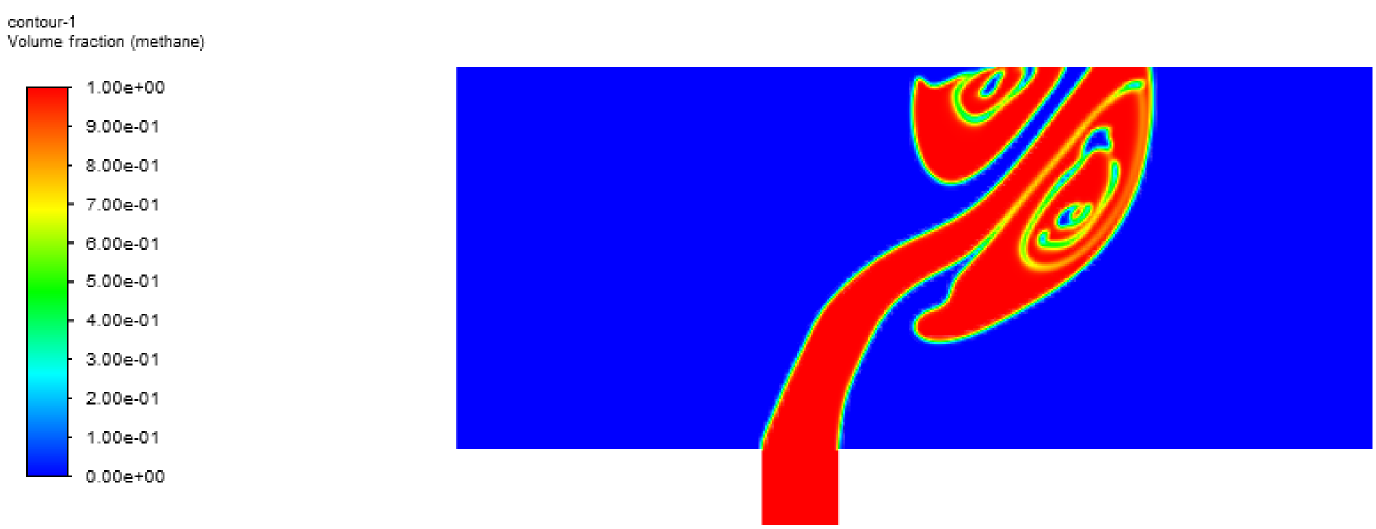

Figure 14: Contour plot of the volume fraction of methane at 12 s

Task 4:

To begin the analysis, the geometry for the 2D chamber with two side pipes and a mesh with elements with a quadratic order and size of 0.0025 m were created. Next, an implicit scheme was chosen in the volume of fluid options, and the turbulence k-epsilon model was chosen. The fluids inside of the tank were set to alcohol and air according to the diagram from the problem statement, and the inlet was set as a velocity inlet which is where the water enters the system. The velocity inlet had a prescribed velocity of 1 m/s, and the gauge pressure was set to zero. The pressure outlet had zero gauge pressure and the backflow phase was set to air. The simulation ran for 400 time steps and a time step size of 0.001 s. As the water hits the alcohol in the tank, the two fluids do not mix, and the alcohol is displaced.

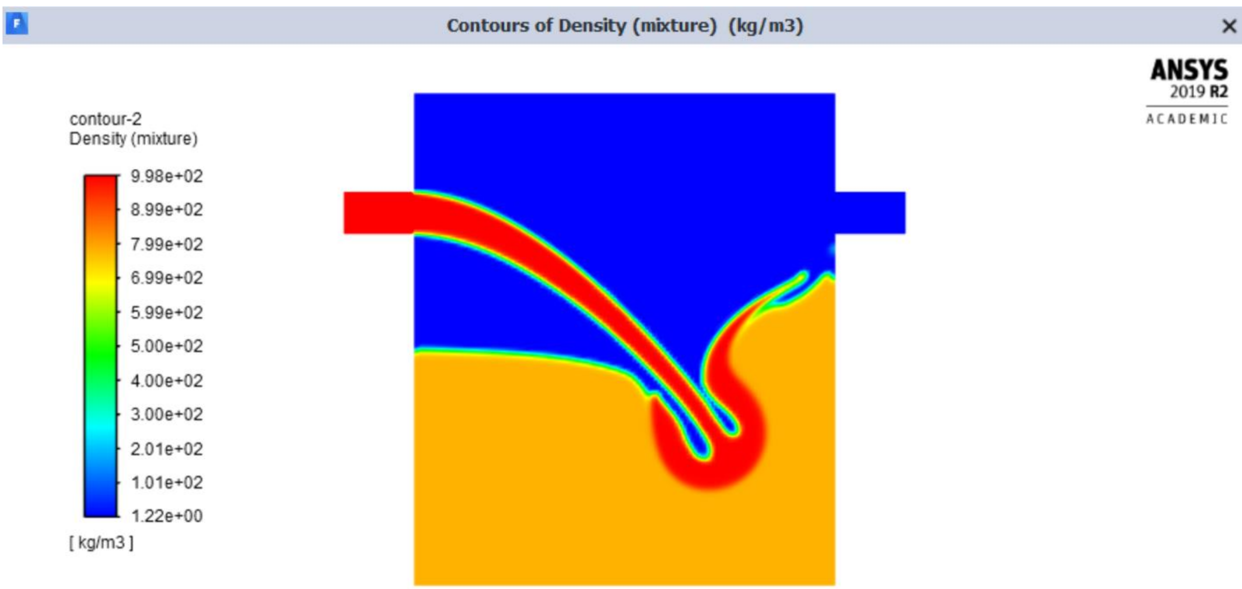

Figure 15: Contour plot of the density of the mixture at 0.4 s