MAE 560: Project 2

Cristina Luna

November 13, 2020

Huei-Ping Huang

Name of Collaborator: David Valderama	
Task(s), Specific Detail	**Contribution to Collaborative Effort**
Task 1	Discussed final conclusions with collaborator
Task 2	Discussed final conclusions with collaborator
Task 3	Discussed final conclusions with collaborator
Task 4	Discussed final conclusions with collaborator

Task 1a:

Background and Approach:

This task simulates the leaking of natural gas from an underground vault into open air, in a pure 2-D setting. This underground vault has its lateral and top boundaries open to air, and its side pipe is connected to a pressurized reservoir of methane. Running a transient simulation and using a pressure-based solver, this simulation included turning on gravity and setting it equal to $-9.81\ m/s^2$ in the vertical “y direction”; using a turbulence k-epsilon model; ignoring surface tension; setting the operating density method to mixture-averaged; setting the underground vault’s lateral and top boundaries 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 $1 * 10^{-5} m^2 s^{-2}$, and turbulence dissipation rate to $1 * 10^{-6} m^2 s^{-3}$. This transient simulation was performed for $t = 7s$.

Deliverables:

A fine mesh resolution was sought to be able to capture accurate simulation values. This consisted of using a mesh size of 0.5 m. To compliment this fine mesh resolution to be able to capture accurate simulation results, the time step size that was used was 0.01 s, and the number of time steps was 700 with 10 max iterations/time step. Using this fine mesh resolution and a small time step size, a contour plot of the volume fraction of methane (phase 2) is shown in the figure below.

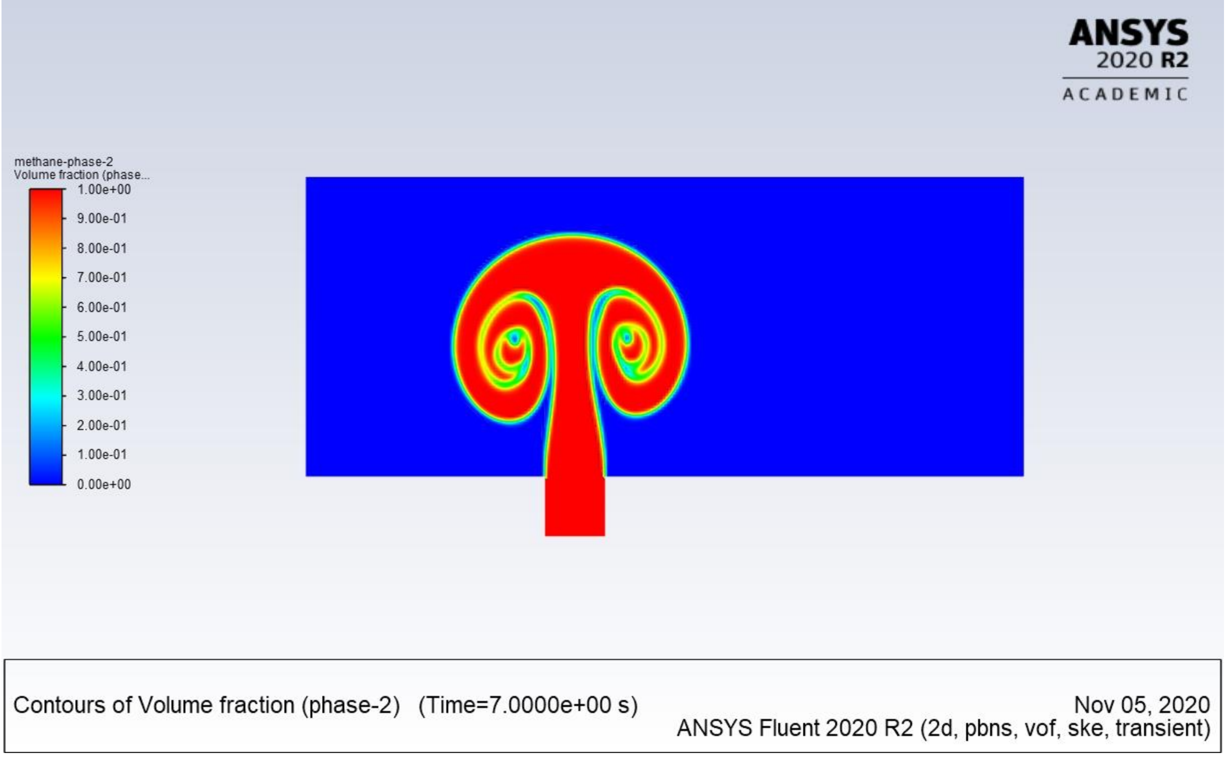


Figure 1: Volume Fraction Contour Methane (Phase 2) at $t = 7\ s$

Task 1b:

Background and Approach:

This task is similar to Task 1a but has one key difference. This task also simulates the leaking of natural gas from an underground vault into open air, in a pure 2-D setting. This underground vault has its lateral and top boundaries open to air, and its side pipe is connected to a pressurized reservoir of methane. 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. Running a transient simulation and using a pressure-based solver, this simulation included turning on gravity and setting it equal to $-9.81\ m/s^2$ in the vertical "y direction"; using a turbulence k-epsilon model; ignoring surface tension; setting the operating density method to mixture-averaged; setting the underground vault's right lateral and top boundary 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, a and y velocity = 0 m/s, phase 2 (methane) volume fraction to 0, turbulence kinetic energy to $1 * 10^{-5} m^2 s^{-2}$, and turbulence dissipation rate to $1 * 10^{-6} m^2 s^{-3}$. This transient simulation was performed for $t = 7s$.

Deliverables:

A fine mesh resolution was sought to be able to capture accurate simulation values. This consisted of using a mesh size of 0.5 m. To compliment this fine mesh resolution to be able to capture accurate simulation results, the time step size that was used was 0.01 s, and the number of time steps was 700 with 10 max iterations/time step. Using this fine mesh resolution and a small time step size, a contour plot of the volume fraction of methane (phase 2) is shown in the figure below.

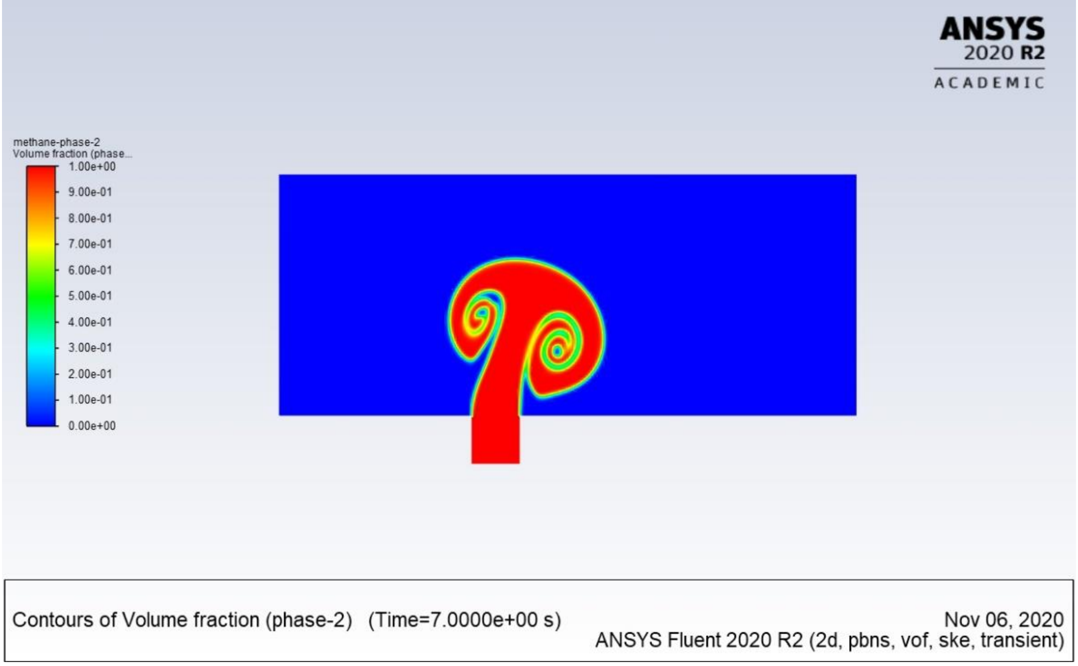


Figure 2: Volume Fraction Contour Methane (Phase 2) at $t = 7\ s$

Task 2:

Background and Approach:

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 20 cm in the square bucket, and a circular region of water, with a diameter of 5 cm, is placed 20 cm above the water surface. Running a transient simulation and using a pressure-based solver, this simulation included turning on gravity and setting it equal to $-9.81\ m/s^2$ in 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 $t = 0.5\ s$.

Deliverables:

A fine mesh resolution was sought to be able to capture accurate simulation values. This consisted of using a mesh size of 0.5 cm. To compliment this fine mesh resolution to be able to capture accurate simulation results, the time step size that was used was 0.001 s, and the number of time steps was 500 with 10 max iterations/time step. Using this fine mesh resolution and small time step size, a contour plot of the volume fraction of water (phase 2) at $t = 0.2\ s$, $t = 0.3s$, $t = 0.5s$; and a contour plot of velocity magnitude at $t = 0.5\ s$ is shown in the figures below.

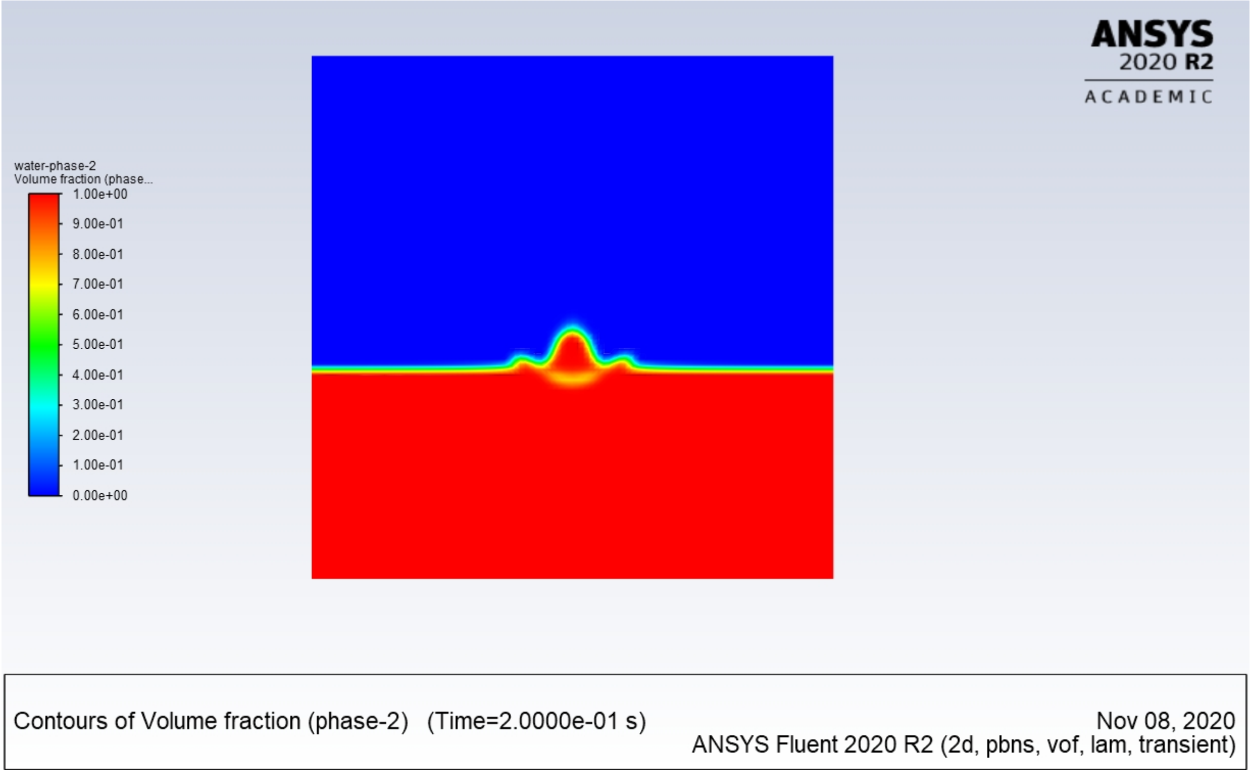


Figure 3: Volume Fraction Contour Water (Phase 2) at $t = 0.2\ s$

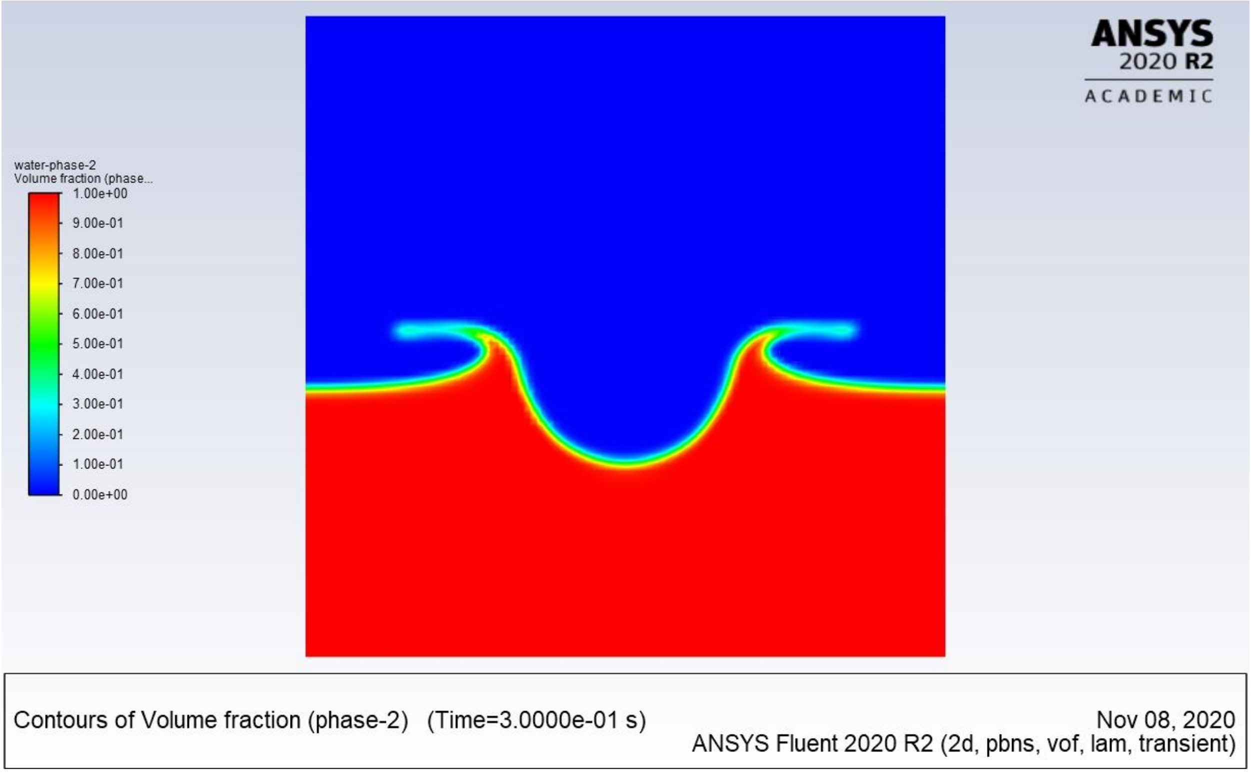


Figure 4: Volume Fraction Contour Water (Phase 2) at $t = 0.3\ s$

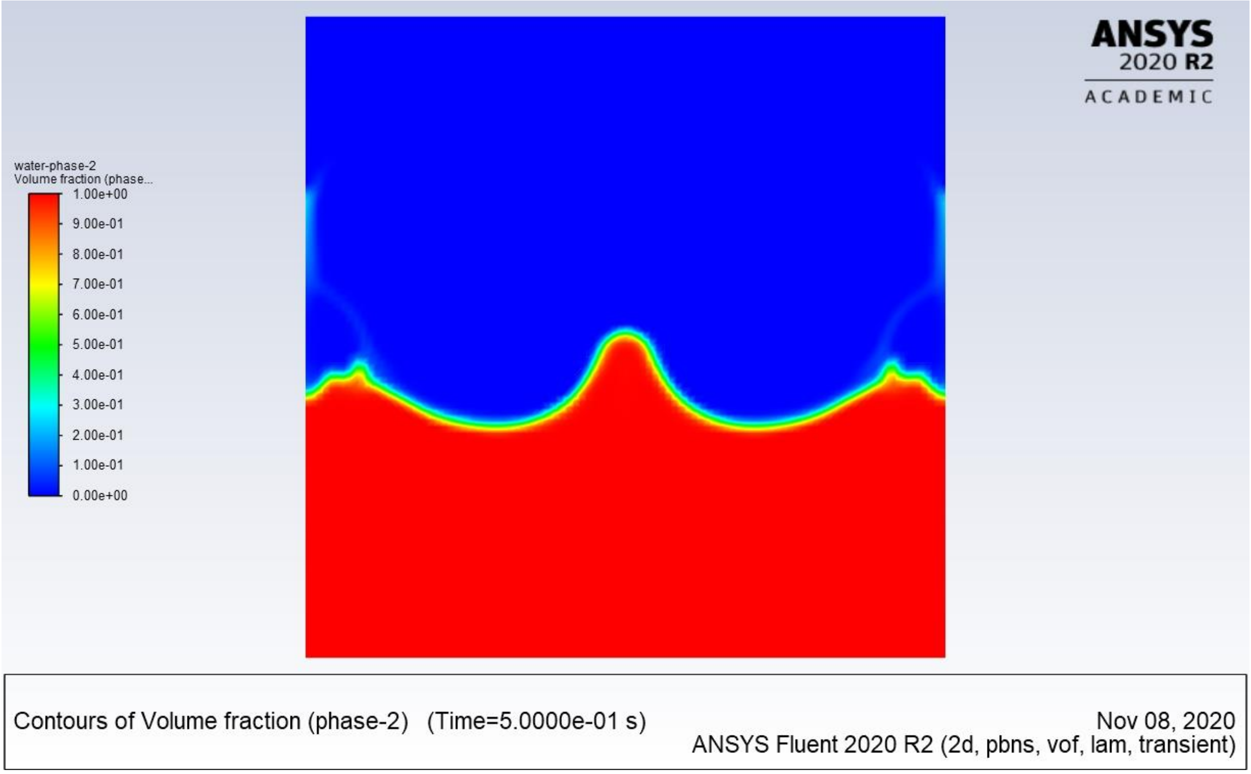


Figure 5: Volume Fraction Contour Water (Phase 2) at $t = 0.5\ s$

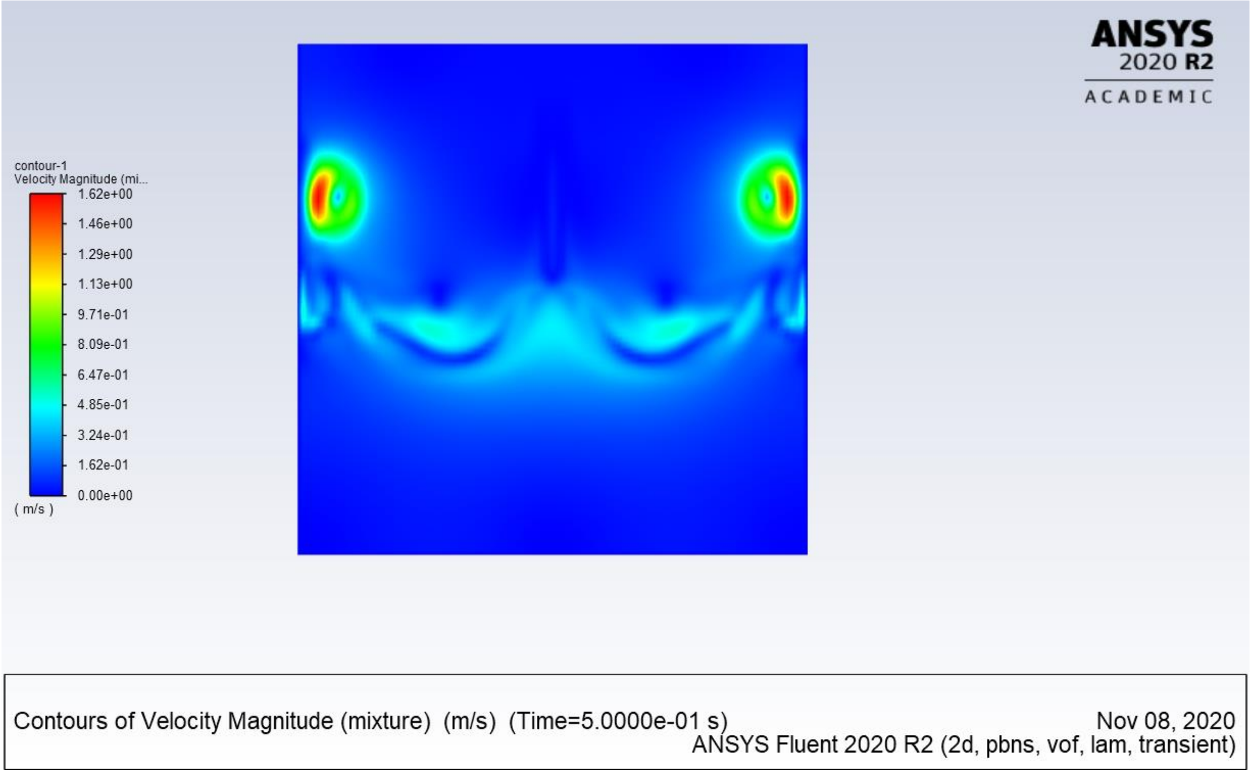


Figure 6: Velocity Magnitude Contour Water (Phase 2) at $t = 0.5\ s$

From Figure 6, it can be seen that the maximum velocities are symmetric about the vertical axis. When the water droplet hits the body of water, it creates a type of splash. This splash, in turn, disturbs the surrounding open air. Because the constant density of air ($\rho = 1.225\ kg/m^3$) is lighter than the constant density of water ($\rho = 998.2\ kg/m^3$), the maximum velocities from Figure 6 is actually the easily disturbed surrounding open air reacting to the splash from the water droplet having hit the body of water.

Task 3:

Background and Approach:

This task simulates the temporal evolution of a glycerin droplet on a $45°$ inclined plate, in a pure 3-D system. This $45°$ inclined plate is surrounded by open air. A hemisphere region, with a radius of 1.5 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 $-9.81\ m/s^2$ in the vertical “y direction”; using a laminar model; turning on surface tension and setting it equal to 0.06 N/m; initializing the system with gauge pressure = 0 Pa, x and y velocity = 0 m/s, and phase 2 (glycerin) volume fraction to 0. This transient simulation was performed for $t = 0.1s$.

Deliverables:

The computational domain was created in such a way that the main process to be simulated will only be affected minimally. To do this, a smaller computational domain was created to be able to seek a fine mesh resolution without reaching the element limit on the Student Version of ANSYS. A 10 cm x 10 cm plate with a 2 cm thickness was created. From Figure 7, the glycerin hemisphere was placed at the center of the plate along the z-axis and 3 cm, in the direction of the +x-axis, from the plate’s left-most side. Because this plate was built on a level surface, gravity in the x-direction was set to $g_x = 9.81 * \sin(45°)$ and $g_y = -9.81 * \cos(45°)$. Furthermore, a fine mesh resolution was sought to be able to capture accurate simulation values. This consisted of using a mesh size of 0.1 cm. To compliment this fine mesh resolution to be able to capture accurate simulation results, the time step size that was used was 0.001 s, and the number of time steps was 100 with 20 max iterations/time step. Furthermore, boundary conditions that were set included setting the plate’s bottom as a wall and all the other sides (top, front, back, left, and right sides) as a pressure outlet with zero gauge pressure and with backflow phase of phase 2 (water) set to 0 to signify that these sides were open to air. Using this computational domain, boundary conditions, mesh resolution, and time step size, three iso-surface (VF = 0.9) contour plots showing the 3-D shape of the glycerin (phase 2) hemisphere at $t = 0\ s$, $t = 0.05\ s$, and $t = 0.1\ s$ are shown in the figures below. Additionally, three contour plots of the volume fraction of glycerin (phase 2) on the plane of symmetry at $t = 0\ s$, $t = 0.05\ s$, and $t = 0.1\ s$ are also shown in the figures below.

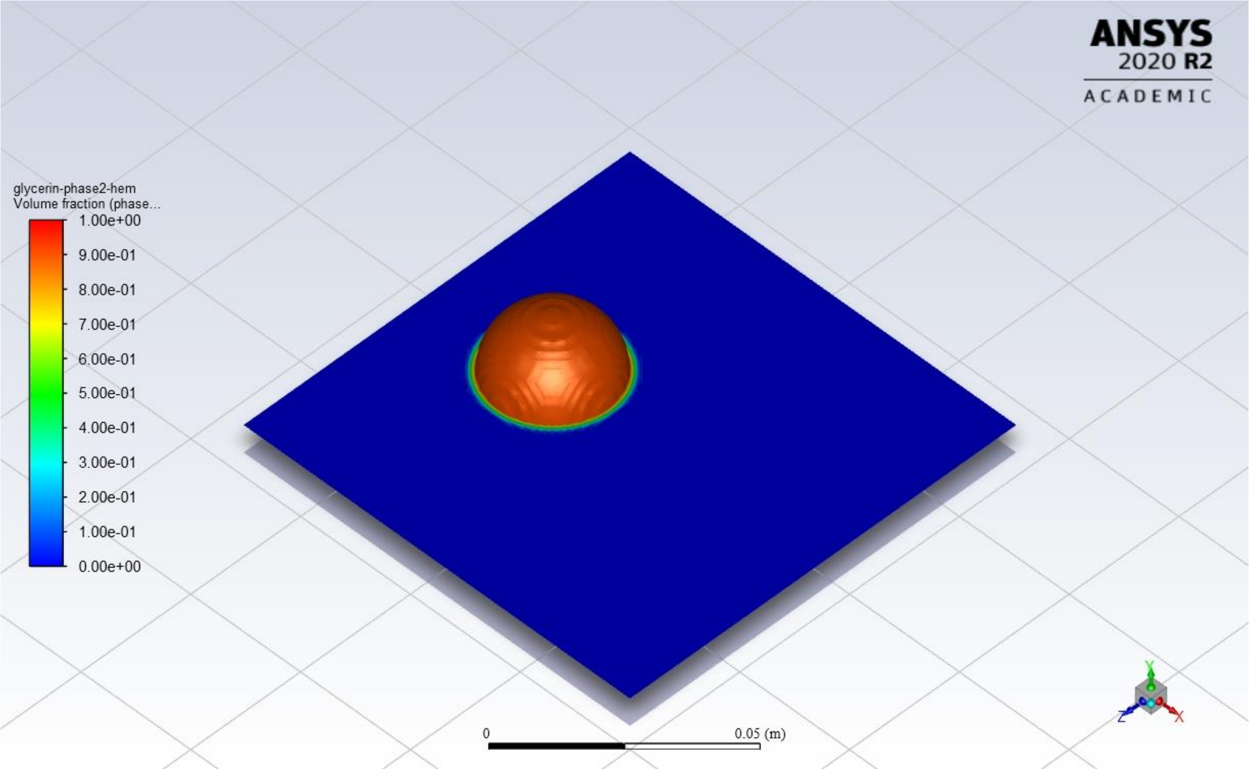


Figure 7: Iso-Surface of 3-D hemisphere of glycerin (phase 2) at $t = 0\ s$

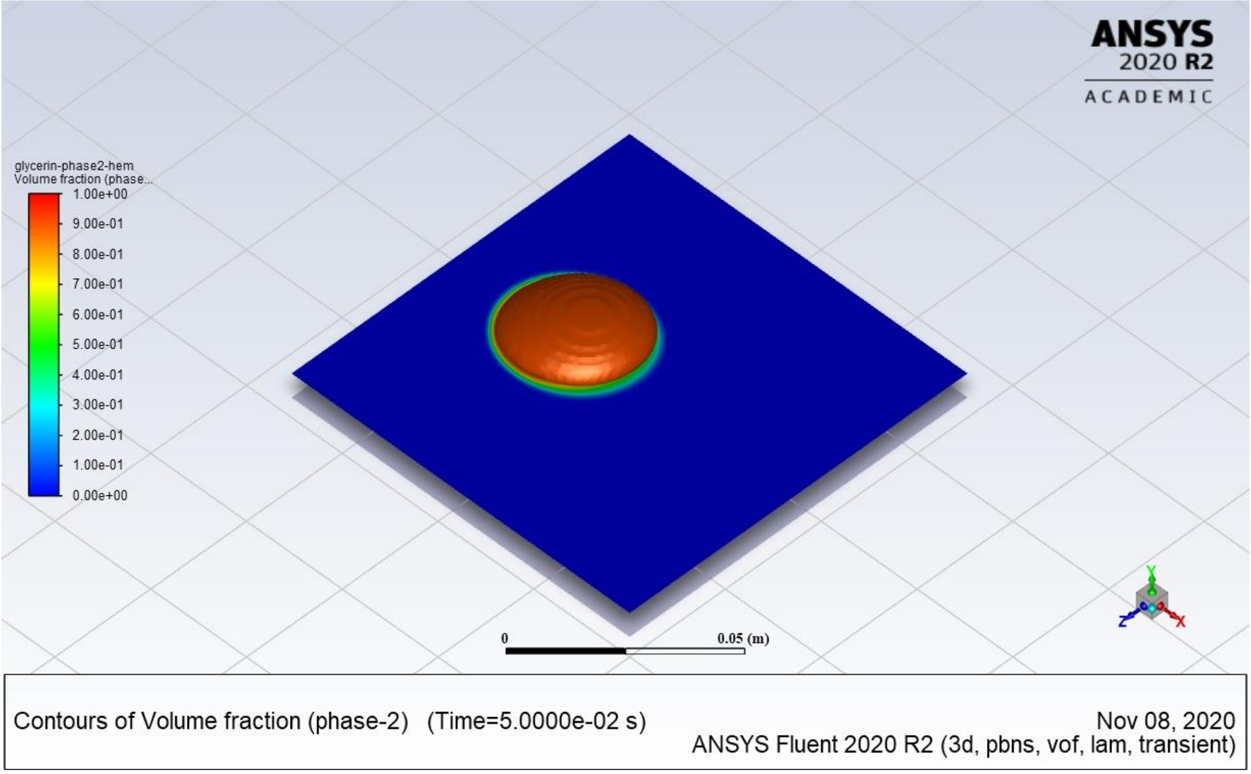
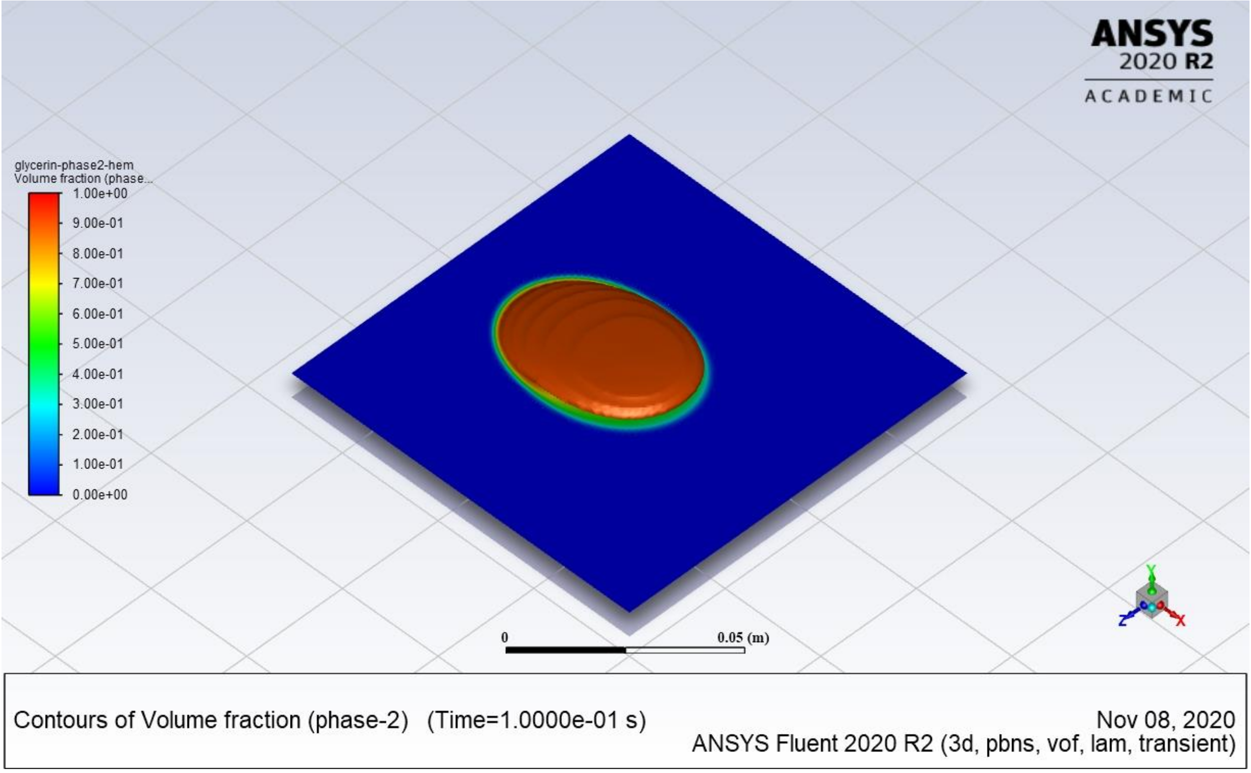


Figure 8: Iso-Surface of 3-D hemisphere of glycerin (phase 2) at $t = 0.05\ s$



Contours of Volume fraction (phase-2) (Time=1.0000e-01 s)
Nov 08, 2020
ANSYS Fluent 2020 R2 (3d, pbns, vof, lam, transient)

Figure 9: Iso-Surface of 3-D hemisphere of glycerin (phase 2) at $t = 0.1\ s$

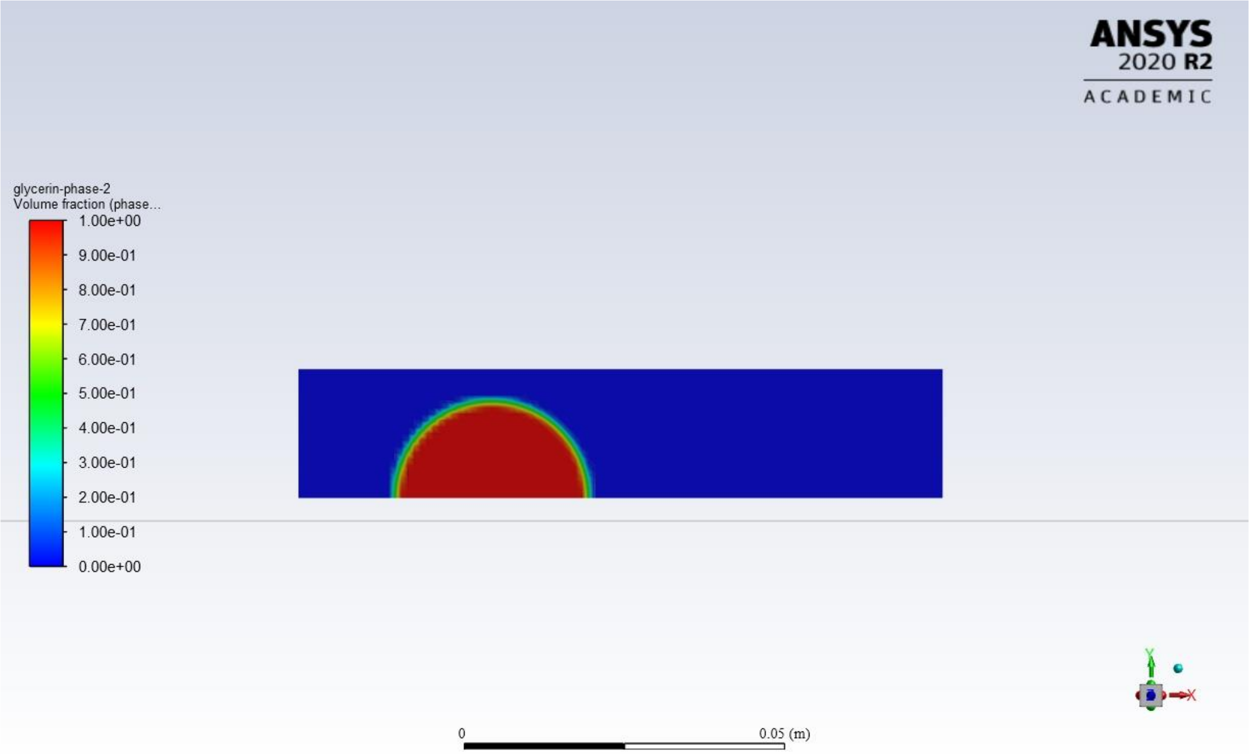


Figure 10: Volume Fraction Contour at the plane of symmetry glycerin (phase 2) at $t = 0\ s$

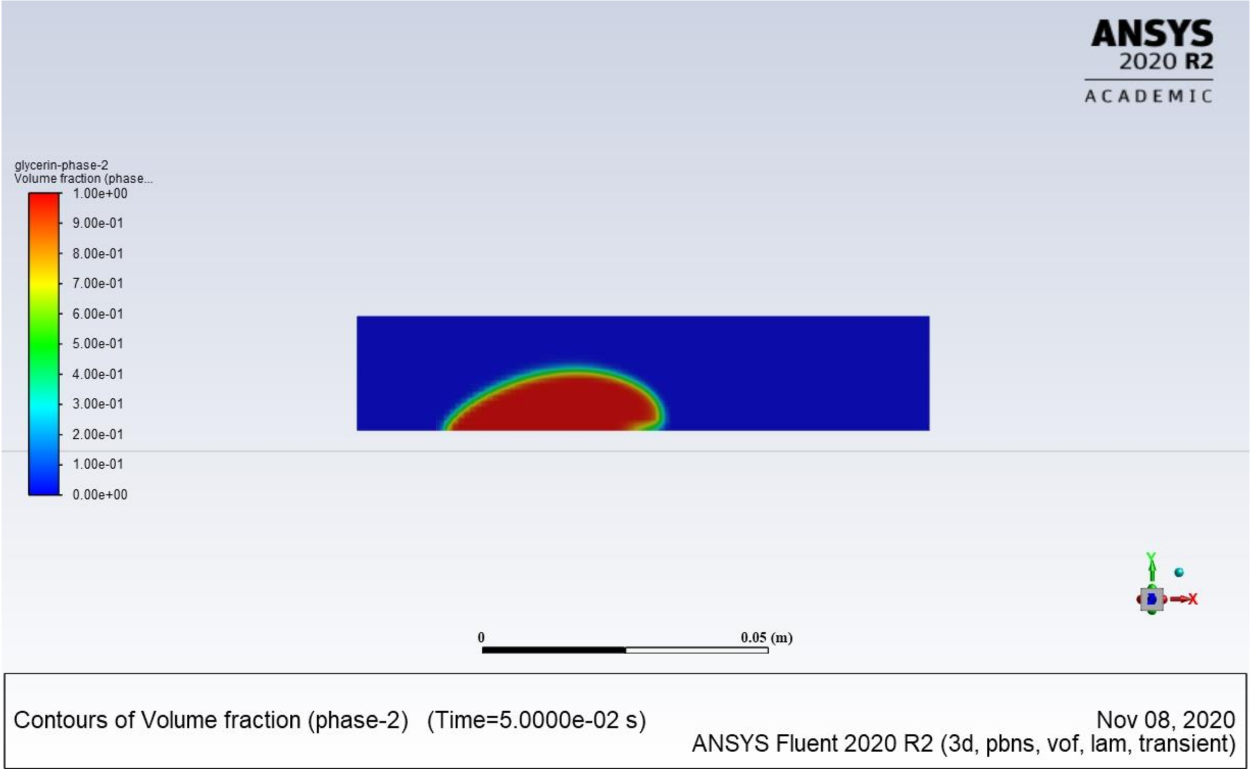


Figure 11: Volume Fraction Contour at the plane of symmetry glycerin (phase 2) at $t = 0.05\ s$

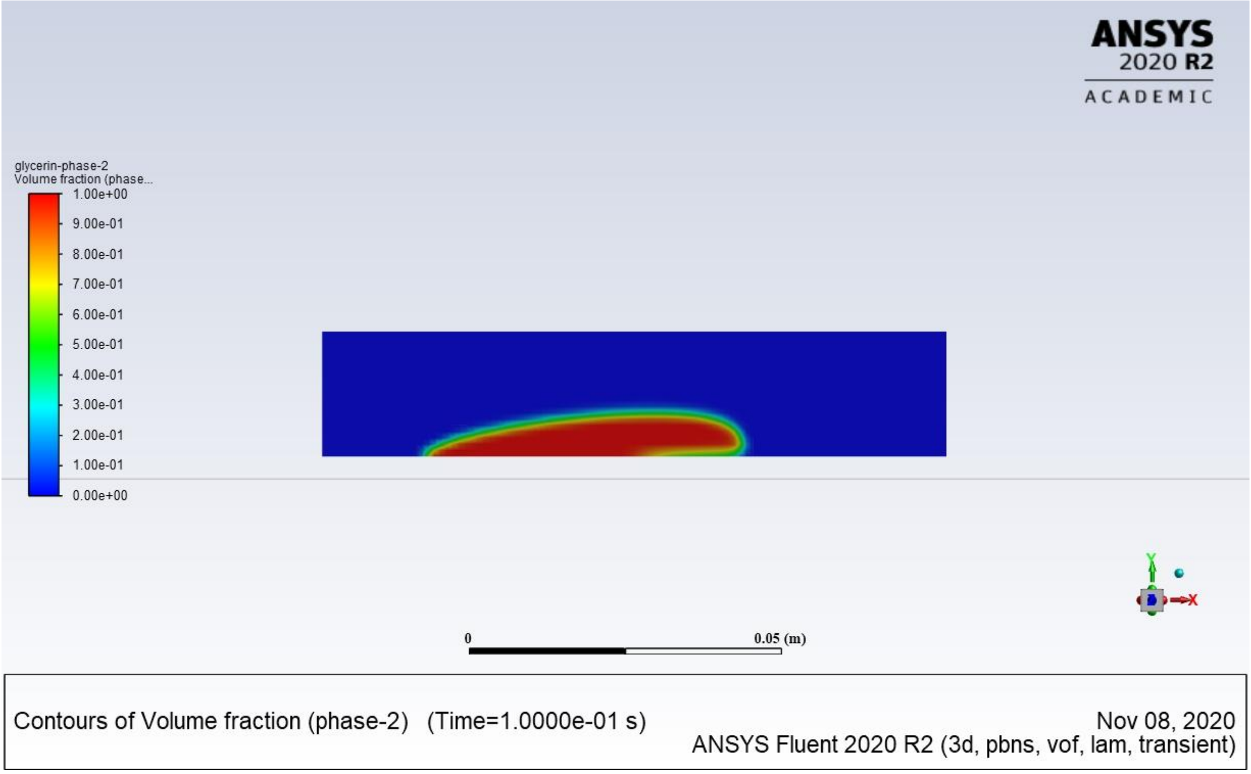


Figure 12: Volume Fraction Contour at the plane of symmetry glycerin (phase 2) at $t = 0.1\ s$

Task 4:

Background and Approach:

This task simulates the flow of water in a U-shaped 3-D pipe with a circular cross section (radius = 2cm). The U-pipe has two openings into open space with air outside. Running a transient simulation and using a pressure-based solver, this simulation included turning on gravity and setting it equal to $-9.81\ m/s^2$ in the vertical "y direction"; using a laminar model; turning off surface tension to eliminate some ambiguities in the definition of "h" in the deliverables; initializing the system with gauge pressure = 0 Pa, x and y velocity = 0 m/s, and phase 2 (water) volume fraction to 1. The system is initialized with water (phase 2) so that the tube can be filled with the other fluid (air) much easier. One cylindrical region of air was created to partially fill the left side leg of the U-shaped 3-D pipe with a 5 cm depth of air, measured from the top. Similarly, another cylindrical region of air was created to partially fill the right side leg of the U-shaped 3-D pipe with a 15 cm depth of air, measured from the top. This transient simulation was performed for $t = 1.25\ s$.

Deliverables:

This simulation was performed using the full-pipe 3-D geometry. The boundary condition that was chosen for the two top openings was that of a pressure outlet, with zero gauge pressure and with backflow phase of phase 2 (water) equal to 0. Both top openings were set as a pressure outlet to properly simulate the oscillation by allowing the air to freely go in and out of the openings. Furthermore, a fine mesh resolution was sought to be able to capture accurate simulation values. This consisted of using a mesh size of 0.3 cm with the inflation set to program controlled. To compliment this fine mesh resolution to be able to capture accurate simulation results, the time step size that was used was 0.002 s, and the number of time steps was 625 with 10 max iterations/time step. Using these boundary conditions and mesh resolution, a line-plot of h (the water level in the left leg relative to the equilibrium level) as a function of flow time and contour plots of the x-velocity on the plane of symmetry at approximately 1/4 and 3/4 of the period of oscillation (when h = 0 cm) are shown in the figures below.

From Figure 5 (b) in the project manual, h is defined as the water level in the left leg of the U-Shaped 3-D pipe relative to the equilibrium level. Since the equilibrium level is defined as 10 cm from the top of the U-Shaped 3-D pipe, at $t = 0\ s,\ h = 5\ cm$. In order to calculate h, a report definition was made of the mass flow rate over the left side pipe's cross section area. Because the formula for mass flow rate is $\dot{m} = \rho V A$, and because the left side pipe's cross section area was defined as having zero gauge pressure with backflow phase of phase 2 (water) equal to 0, the mass flow rate was divided by the constant density of air ($\rho = 1.225\ kg/m^3$) and the constant cross section area ($A = \pi(0.02\ m)^2$) to obtain a plot of the system's velocity vs. flow time. Since h is a position, and because the time step size used was relatively small, the data was integrated using the trapezoidal method to obtain a plot of h vs. flow time. Since a decrease/increase in h equals to an increase/decrease in mass flow rate, respectively, and because the mass flow rate was initially positive, the plot of h vs. flow time was multiplied by a factor of negative 1 and offset by an initial h (water level) of 5 cm to represent h initially being equal to 5 cm at $t = 0\ s$. From the plot below, $t_1\left(\approx \frac{1}{4} of\ oscillation = h = 0\ cm\right) = 0.256\ s$, $t_2\left(\approx \frac{3}{4} of\ oscillation = h = 0\ cm\right) = 0.772\ s$, and $period\ (1\ oscillation) = 1.026\ s$.

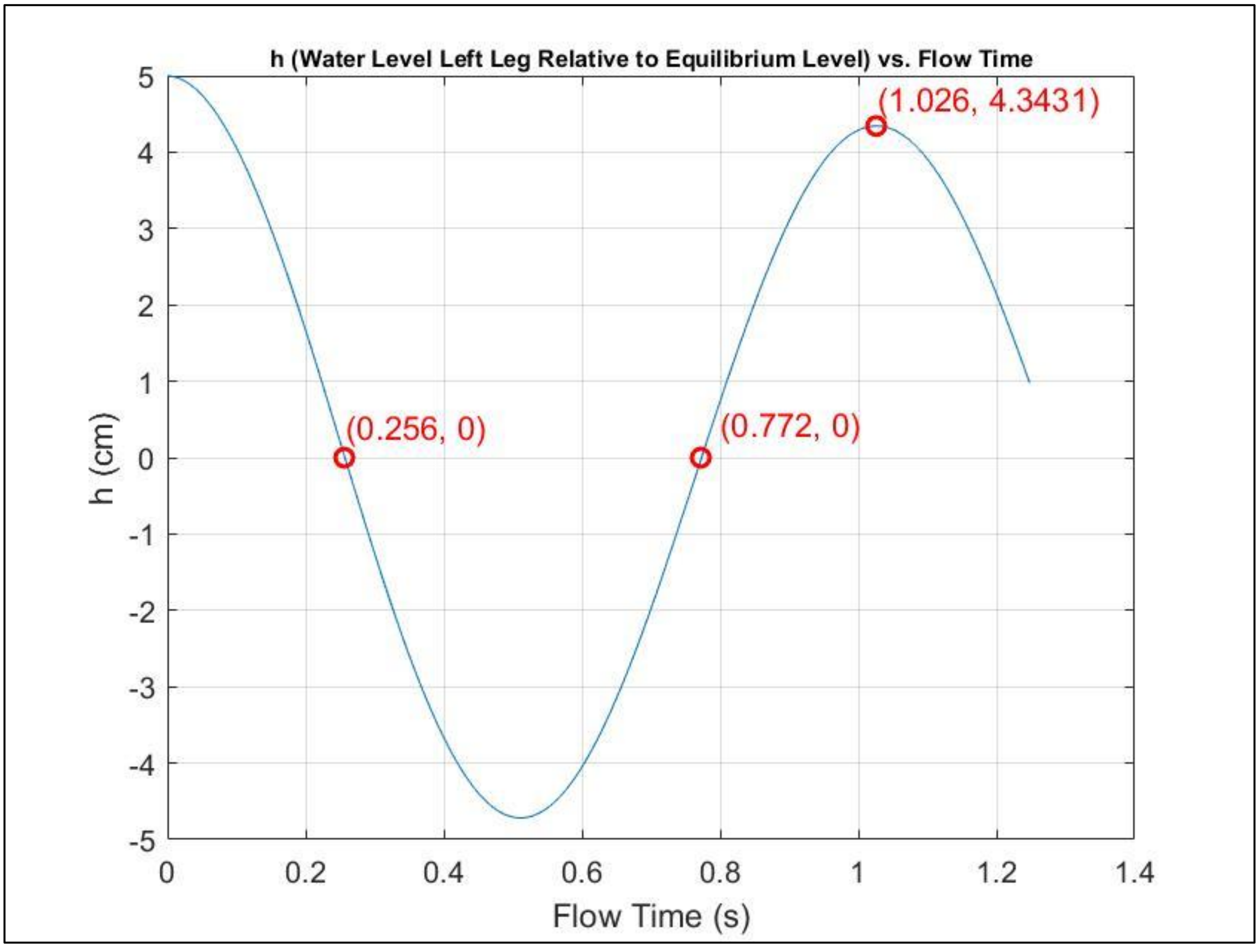


Figure 13: Line plot of h as a function of time

At $t_1 \left(\approx \frac{1}{4} of\ oscillation \approx h = 0\ cm\right) = 0.256\ s$, the contour plot of the x-velocity on the plane of symmetry is shown in the figure below.

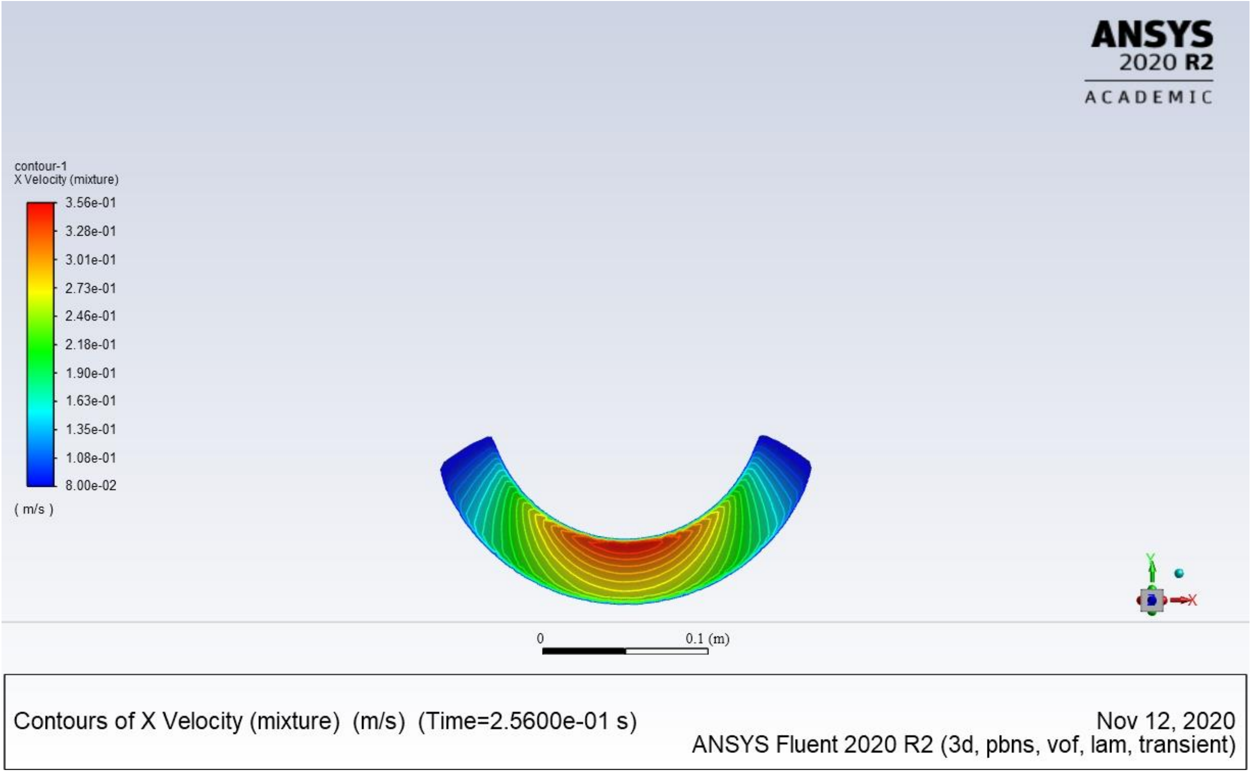


Figure 14: X-Velocity Contour Plot at $t_1 \left(\approx \frac{1}{4} of\ oscillation \approx h = 0\ cm\right) = 0.256\ s$

At $t_2 \left(\approx \frac{3}{4} of\ oscillation \approx h = 0\ cm\right) = 0.772\ s$, the contour plot of the x-velocity on the plane of symmetry is shown in the figure below.

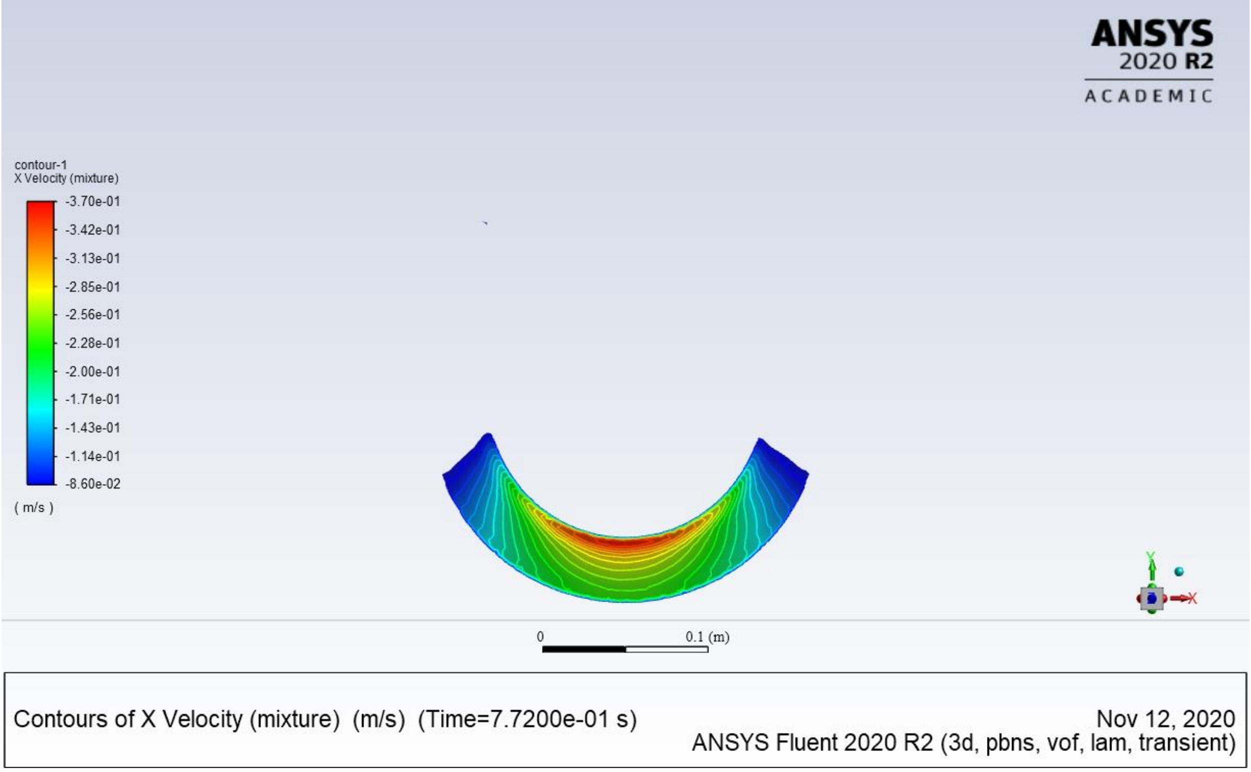


Figure 15: X-Velocity Contour Plot at $t_2 \left(\approx \frac{3}{4} of\ oscillation \approx h = 0\ cm\right) = 0.772\ s$

From Figure 14, it can be seen that at $t_1 \left(\approx \frac{1}{4} of\ oscillation \approx h = 0\ cm\right) = 0.256\ s$, the x-velocity is positive, meaning that the water flowing inside the 3-D U-Pipe is moving from left to right. Because the water inside the 3-D U-Pipe is moving from left to right, h decreases during this time. Alternatively, from Figure 15, it can be seen that at $t_2 \left(\approx \frac{3}{4} of\ oscillation \approx h = 0\ cm\right) = 0.772\ s$, the x-velocity is negative, meaning that the flow water inside the 3-D U-Pipe is moving from right to left. Because the water inside the 3-D U-Pipe is moving from right to left, h increases during this time. The Colormap size for the contour plots in Figure 14 and Figure 15 were adjusted to 20 to allow for more of the features of the contour plots to be displayed. For both Figure 14 and Figure 15, the maximum velocity occurs near the inner edge of the bottom of the pipe. This contrasts the theory that velocity and radius are proportional to each other, allowing for the velocity to increase as its radius from the center does. Discrepancies from factors within the simulation may have led to this discrepancy.