# MAE 598
# APPLIED COMPUTATIONAL FLUID DYNAMICS

Project 3

Richard Nile
(No Collaboration)

November 8, 2018

**Task 1:**

For this task, a simple 2D geometry was used to simulate water faucet behavior over time. The geometry and mesh of the 2D system are shown in Figure 1. The rectangular portion is 22 × 20 cm and the curved faucet has a width of 2 cm. For the initial simulation, gravity was set to -0.1 $m/s^2$ in the y-direction. The VOF model for multiphase flow was used. Implicit formulation of volume fraction parameters was used to increase the stability of calculations. Implicit body force formulation was used to include the effects of gravity. A laminar viscosity model was used based on the size and conditions of the system.

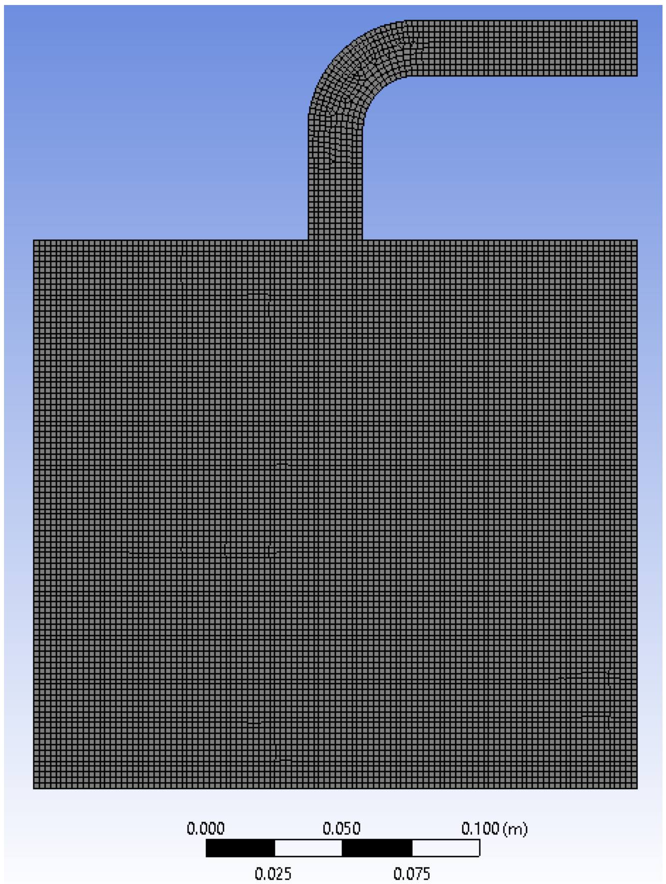


*Figure 1: Geometry and mesh of 2D faucet*

The rectangular region was initially all air and the curved entrance was all water. The density of both phases was assumed constant, and a pressure-based solver was used to find a transient solution. Water entered with a uniform velocity of 0.1 m/s normal to the inlet wall. Figure 2 shows a contour plot of both phases after 0.3 s. The water phase narrows slightly as the fluid is stretched by downward acceleration. The water begins to spread out as it meets the air at a high velocity. Figure 3 shows the simulation after 1.0 s. The water profile further develops as gravity increases the downward speed. The air resistance caused the water to “fan out” as it moves downward.

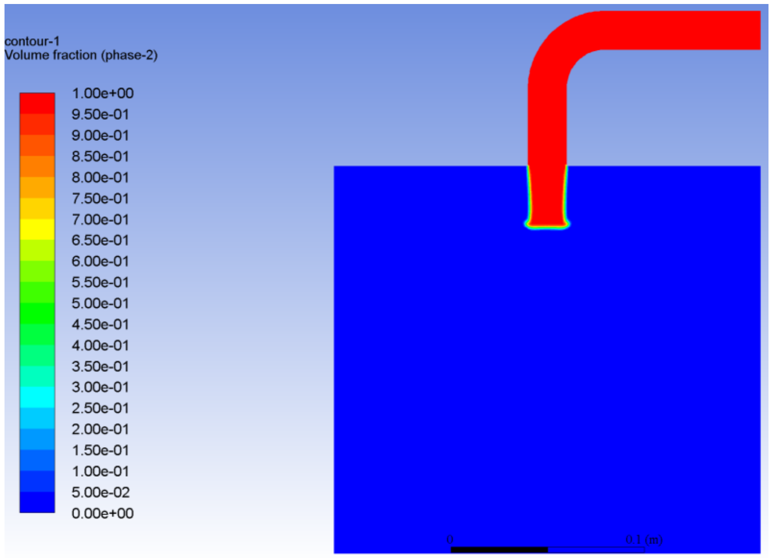


*Figure 2: Contour plot of volume fraction after 0.3 s*

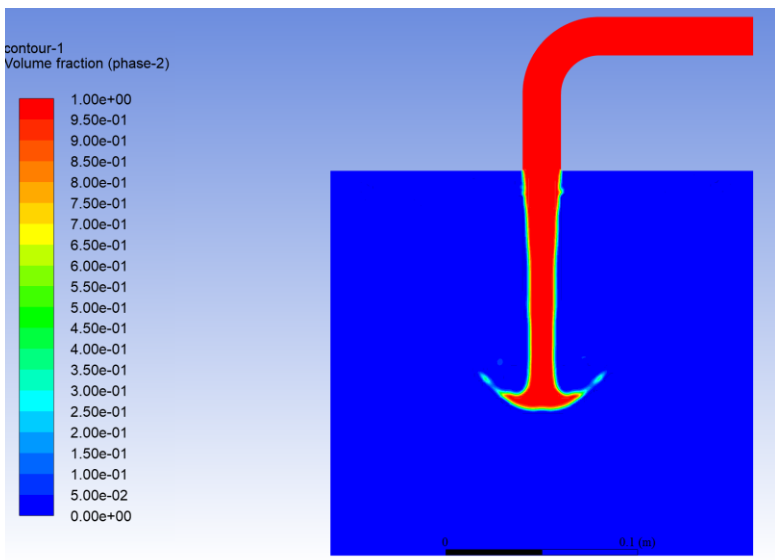


*Figure 3: Contour plot of volume fraction after 1.0 s*

Another simulation was run with the same conditions except surface tension was included. The value of the surface tension of water was assumed constant at 0.0719 N/m which is accurate for these phases. To compare the effects of surface tension, Figures 4 and 5 show the volume fraction contour plot at 0.3 and 1.0 s as before. Unlike the water phase in Figure 2, Figure 4 shows that tensional forces are causing a water droplet to form on the faucet. The surface tension forces are stronger than the gravitational force, so a rounded droplet forms but does not fall.

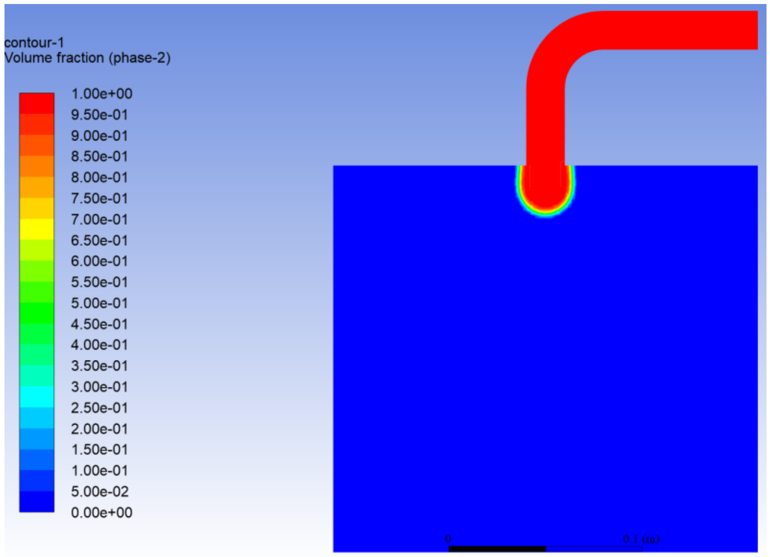


*Figure 4: Contour plot of volume fraction after 0.3 s (with surface tension)*

In Figure 5, the volume of the droplet increases, but it still does not fall. Because gravity was very low, the droplet would have to get larger for gravitational forces to overcome surface tension forces. Gravity did cause the smaller droplet width at the top because it was essentially stretching the droplet. The asymmetry seen in the first simulation was not present in this simulation because strong surface tension forces form the water into a rounded shape.

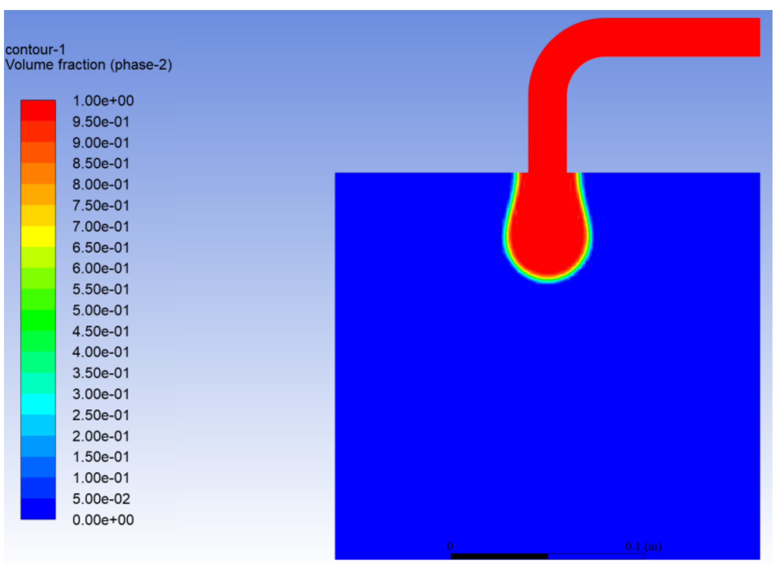


*Figure 5: Contour plot of volume fraction after 1.0 s (with surface tension)*

The final simulation with this geometry was to model a methane gas leak. The same geometry was used, but it was scaled up by a factor of 100. The direction of gravity was reversed, and it was increased to 9.81 m/s$^2$. The k-epsilon turbulence was deemed most appropriate for the size of the system. As before, the rectangular section was initially filled with air. This time, the curved section was filled with methane. Methane entered the pipe with a uniform constant velocity of 5 m/s.

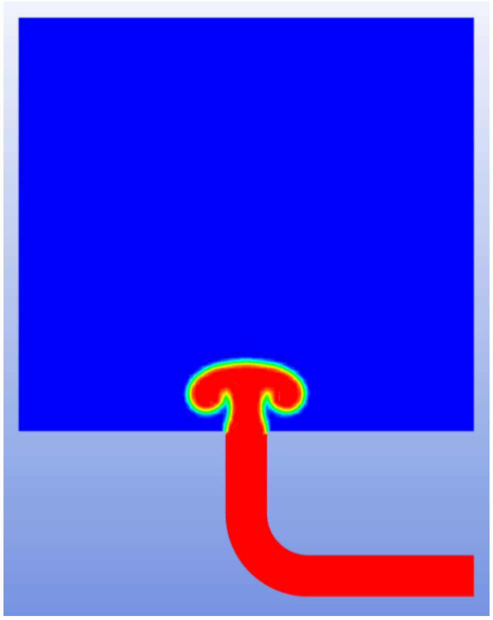

*Figure 6: Contour plot of volume fraction after 1.5 s*

Figure 6 shows the contour plot of the simulation at 1.5 s. The methane phase is shown in red. After leaving the pipe, the methane rose in air because of its lower density. It also had residual momentum from being forced out of the pipe. The size of the system means turbulent viscosity model was applicable. As the methane moved past the stationary air, it slowed at the sides. This caused a “mushroom cloud” to develop where the methane spun in circles around the central rising column. This was consistent with real gas behavior seen on a large scale (e.g. explosions). Figure 7 shows the system after more time. The “mushroom cloud” shape further developed as turbulence causes gradual mixing of the two phases. Despite the complex shape of the methane phase, it all gradually rose because of the difference in density.

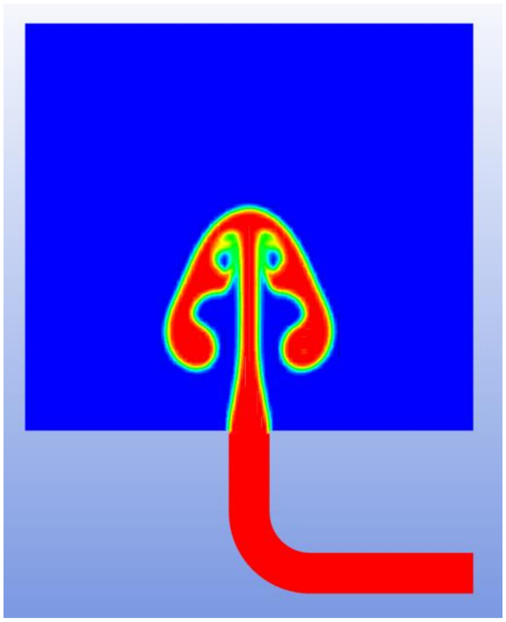

*Figure 7: Contour plot of volume fraction after 4.0 s*

**Task 2:**

This task simulates a glycerin drop sliding down a 15° ramp. The domain of the simulation was a rectangular prism with a width of 15 cm, length of 25 cm, and height of 10 cm. Only the bottom side of the rectangular domain was set to a wall condition. The other five sides were defined as pressure outlets. A glycerin hemisphere was initially at a symmetrical position at the upper end of the ramp. To simulate the tilt of the ramp, the gravity in the y-direction was set to -9.476 $m/s^2$ (-9.81cos15°) and the x-direction to 2.539 $m/s^2$ (9.81sin15°). Figure 8 shows the initial position and shape of the blob on the bottom surface.

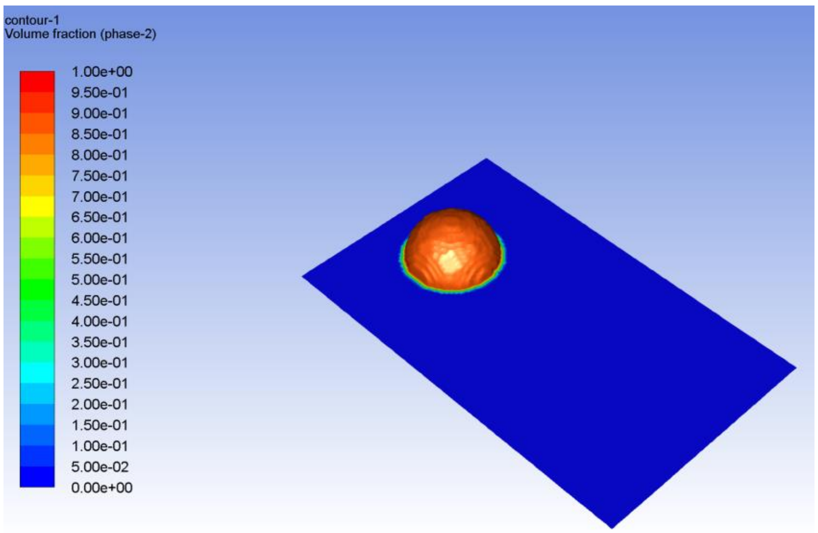


*Figure 8: 3D iso-surface plot of glycerin blob at 0 s*

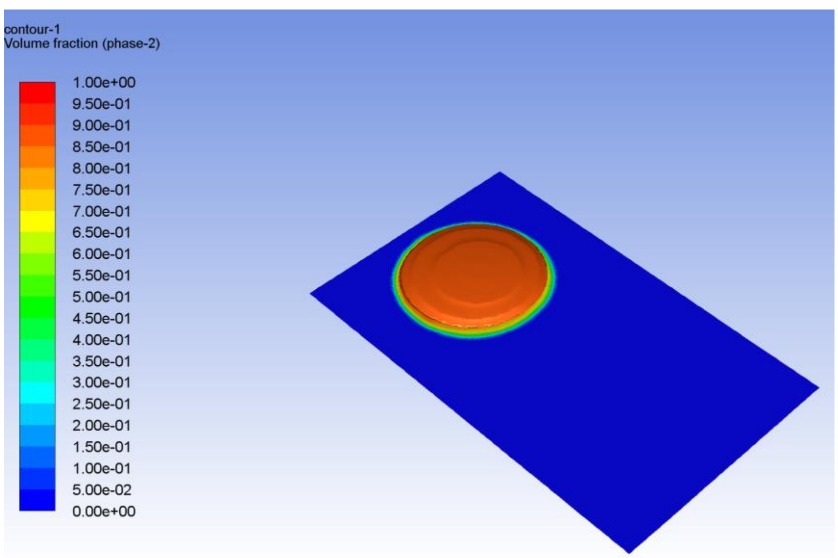


*Figure 9: 3D iso-surface plot of glycerin blob at 0.1 s*

Over time, the blob quickly flattened as the force of gravity was mostly normal to the bottom surface. Figure 10 shows the blob after another 0.1 s had passed. The blob continued to flatten and began to slide down the ramp. The gravitational force in the direction parallel to the bottom plate was almost four times smaller than the gravitational force normal to the plate. This explains why the blob very gradually slid down the ramp. Using a steeper ramp or less viscous material would result in faster sliding.

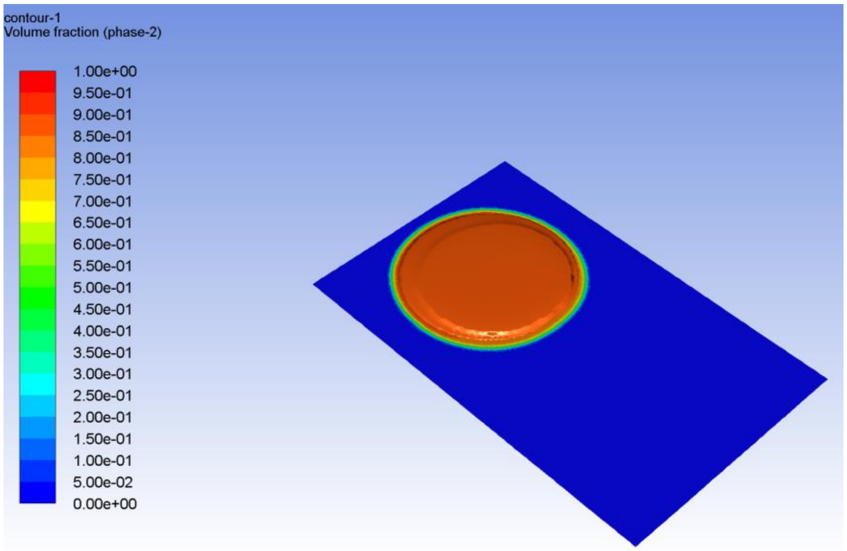


*Figure 10: 3D iso-surface plot of glycerin blob at 0.2 s*

**Task 3:**

The final simulation was a curved pipe like a manometer with an initial liquid level differential. Figure 11 shows the geometry and mesh of the system. The radius of the pipe was 0.05 m and the straight edges are 1.0 m in length. The curved section had a centerline with a radius of 0.5 m. The mesh was enhanced along the wall. Gravity was set to -9.81 m/s$^2$ in the y-direction. The narrow pipe and low anticipated velocities justify the laminar behavior assumption. The left side of the pipe had an initial water level 0.1 m from the top and the right was 0.7 m. The remaining space was air. Figure 12 shows the volume fraction contour plot at the initial state.

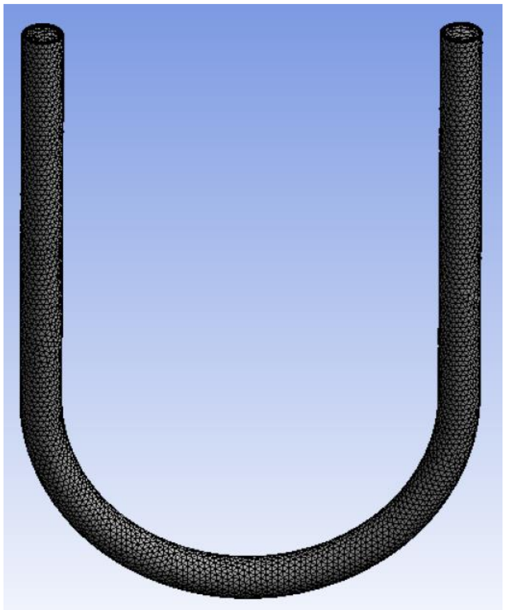

*Figure 11: Curved pipe geometry and mesh*

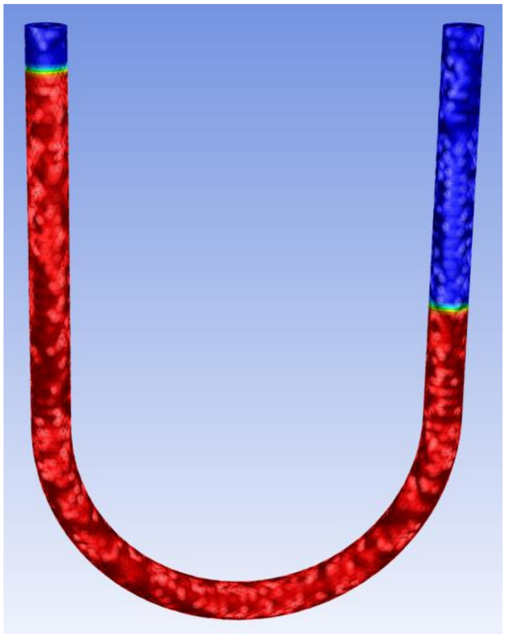

*Figure 12: Volume fraction contour plot at 0 s*

The boundary condition used on both sides was pressure outlet with air backflow only for both ends of the pipe. The simulation was run for 3.0 s using 600 time steps of 0.005 s. The level of water was defined as the average height of water in the left side relative to the equilibrium height. The area-weighted-average x-velocity was calculated with Equation 1. For meaningful results, the area of integration was a cross-section of the pipe where only the water phase was present. For this reason, the horizontal cross-section of the pipe in between the vertical sections was chosen. When this value was positive, water was moving from the left to right and the level.

$$\overline{u_x} = \frac{1}{A} \iint_A u_x dA \tag{1}$$

ANSYS Fluent was used to output this area-weighted-average x-velocity as a function of flow time. Because density was constant, this average velocity corresponds to all cross-sections of water phase. Integration was used to find the change in position of the water-air boundary over time. This value was translated into a more meaningful water level plot shown in Figure 13. As defined, the initial level was 30 cm. The first time the level reached zero was defined as $t_1$ which was at 0.595 s. The level reached a minimum of -28.97 cm at 1.185 s. Note that this value was slightly smaller than -30 cm because of energy losses due to viscosity. The second time the level reached its equilibrium value was defined as $t_2$ at 1.785 s. One cycle was completed after 2.375 s. The left side water level returned to a value of 27.35 cm which was 2.65 cm smaller due to viscous damping.

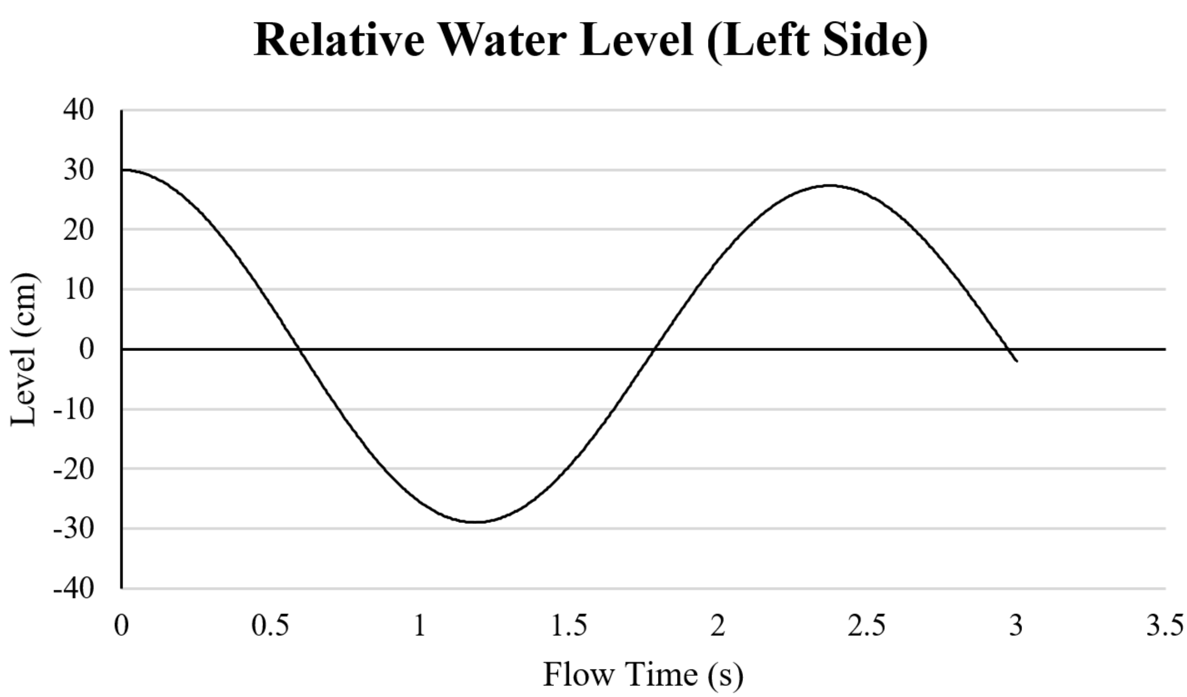


*Figure 13: Left side water level as a function of flow time*

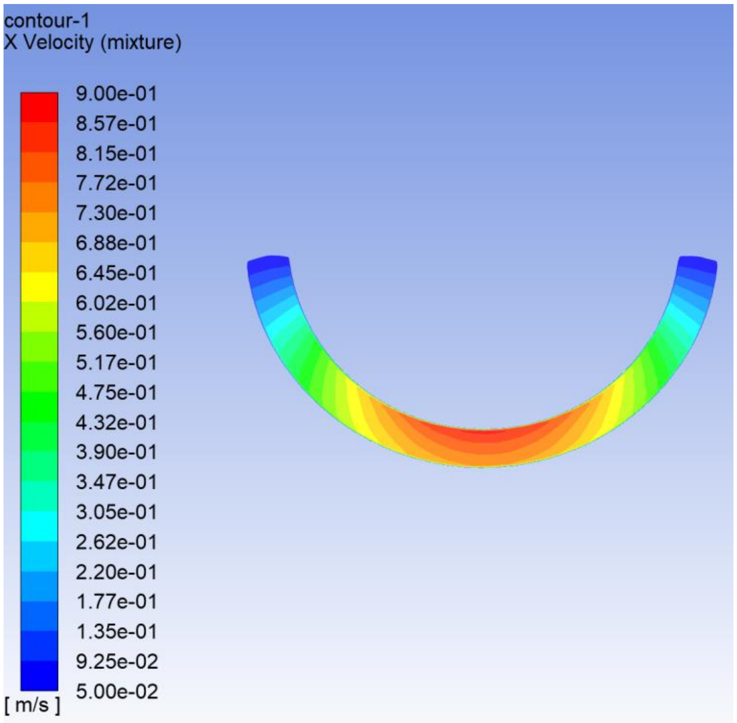


*Figure 14: Contour plot of x-velocity in the plane of symmetry ($t = t_1$)*

Figures 14 and 15 show contour plots of x-velocity at $t_1$ and $t_2$, respectively. The highest x-velocity in both plots occurs in the nearly horizontal sections. The velocity was positive at $t_1$ because water was moving from the left side to the right. As expected the opposite was seen at $t_2$. The average velocity was slightly smaller at $t_2$ due to viscous damping. Flow through curved sections generally produces an asymmetrical velocity profile where the outside edge of the curve generally has the largest velocity.

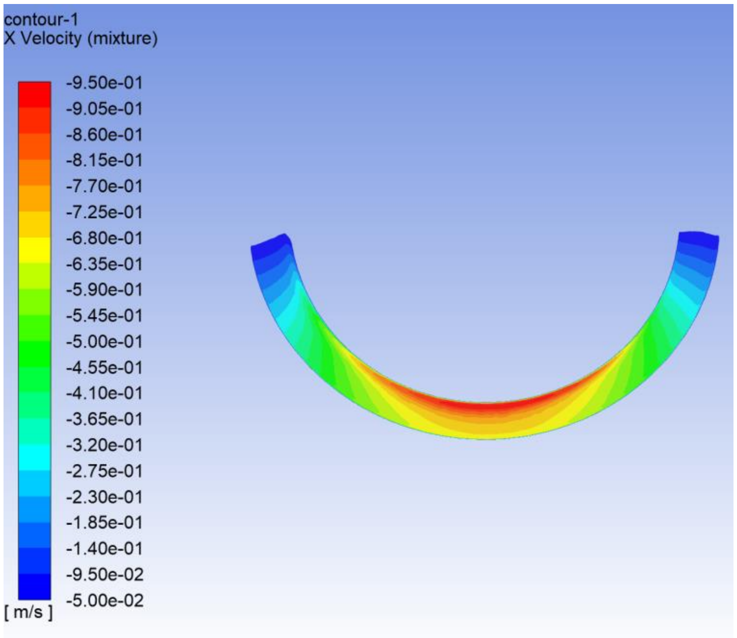


*Figure 15: Contour plot of x-velocity in the plane of symmetry ($t = t_2$)*

However, the opposite was demonstrated in this system at both $t_1$ and $t_2$ because the maximum velocity was near the inner edge. The velocity would be expected to be largest at the outer edge under steady state conditions with flow perpendicular to gravity. These contour plots meet neither of those criteria which explains the apparent discrepancy. The influence of static pressure or the quick cyclical motion of the system were the most likely causes of the large velocity on the inner edge.

The boundary conditions for the two pipe openings were critical for a valid transient solution. It has already been shown that setting them both to pressure outlet with only air as a backflow was successful. Setting the right side opening to wall and leaving the left side alone resulted in no oscillation. Animation of the phase contour plot reveals some minor disturbance at the phase boundary, but the level does not change. Because the simulation assumed incompressible behavior and the air had no outlet on the right, the system was already at its stable state.

Another simulation was attempted leaving the left side as pressure outlet and changing the right side to the default outflow condition. The result was the same as the wall condition. The level does not change because outflow boundary conditions were not compatible with multiphase flow according to ANSYS's user guide. The final combination of boundary conditions was default pressure inlet for both sides. This did produce oscillation completely identical to the oscillation with the original boundary conditions. Figures are not displayed for these conditions because they perfectly match Figures 13-15. For systems where fluid needs to come in and out to maintain continuity, the default pressure inlet or pressure outlet boundary conditions can be used.