Cody Houston

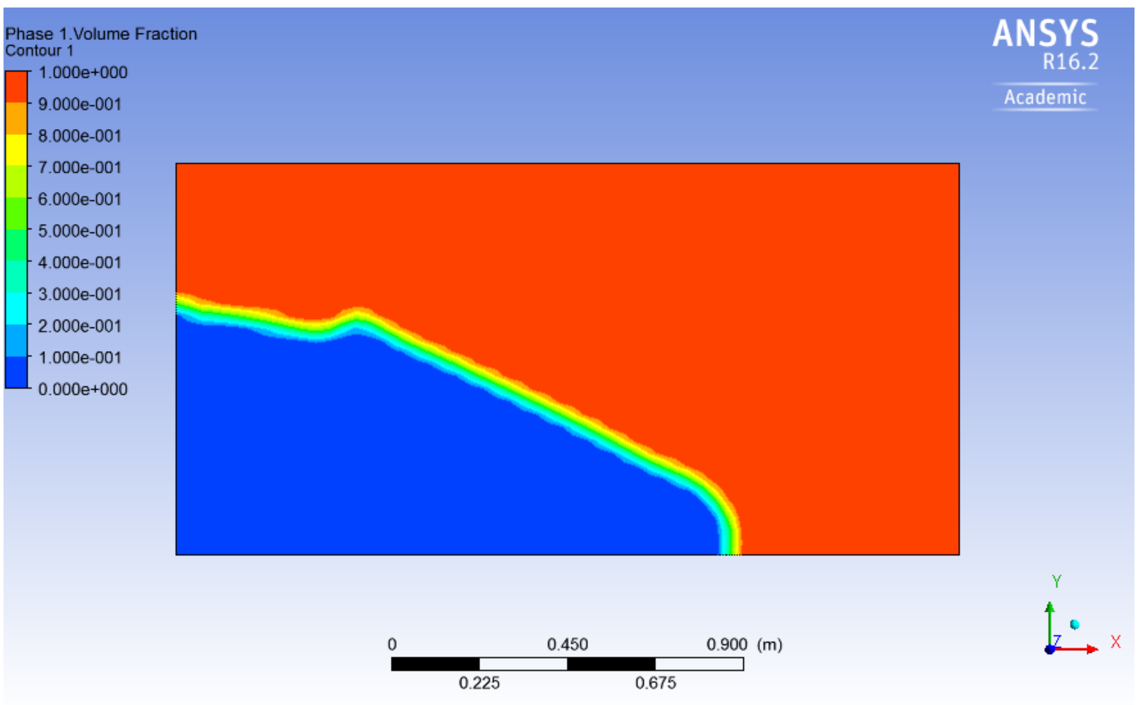
MAE 494

Project 2

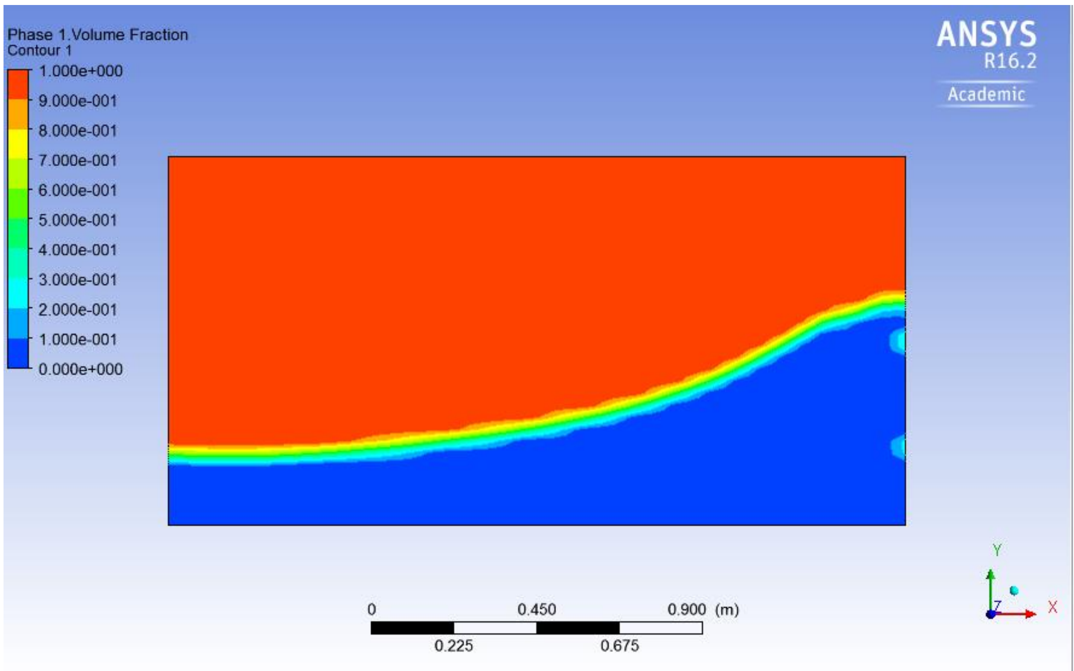
Task 1)

A) This task simulated a small section of water in a tank filled with water, similar to a dam break problem. An analysis was conducted using the turbulent K-epsilon model with default values, and the phases were assigned using mesh marking via the adapt method. The transient analysis was carried out with a time step of .01s, and a max of 15 iterations per step. The solution method was set to PISO, per the directions, and the solution criteria was set to .001 for all fields. This limited the amount of computing time, while delivering results with acceptable resolution. The contour plots of phase 1 volume fraction (Engine oil) can be seen below at various time steps.

T=2s

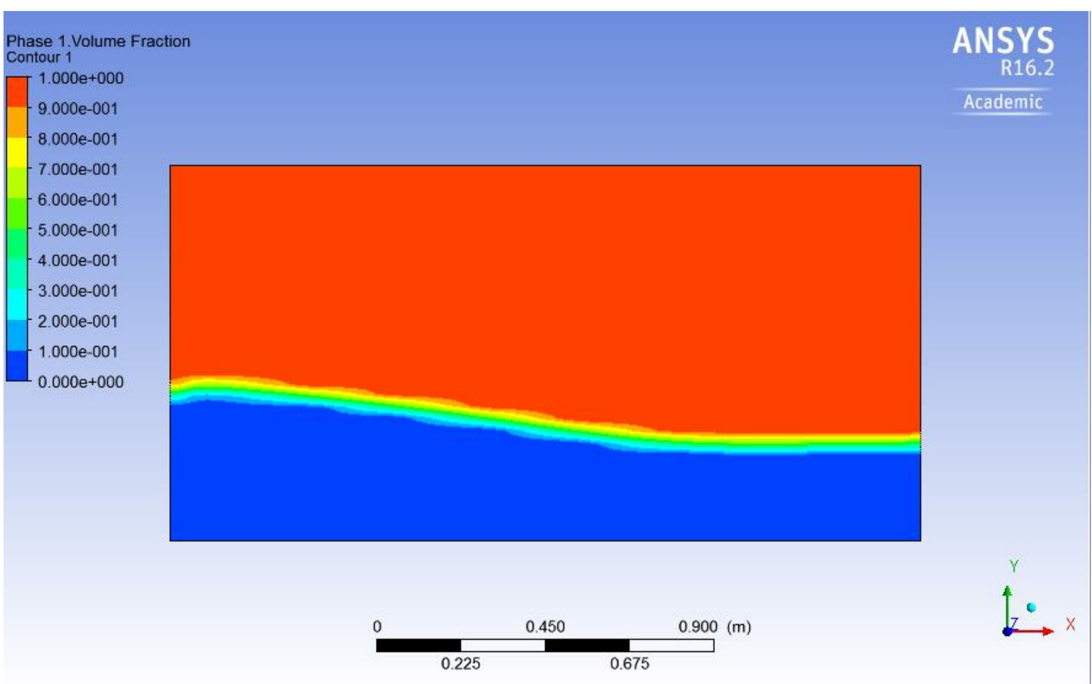


T=6s



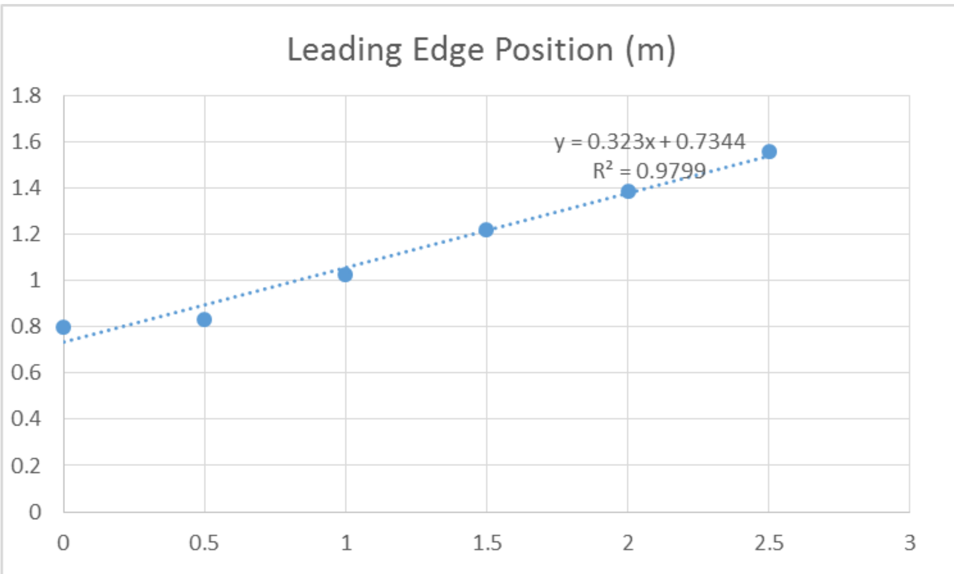
Phase 1.Volume Fraction
Contour 1
1.000e+000
9.000e-001
8.000e-001
7.000e-001
6.000e-001
5.000e-001
4.000e-001
3.000e-001
2.000e-001
1.000e-001
0.000e+000
ANSYS
R16.2
Academic
0
0.450
0.900 (m)
0.225
0.675
Y
X

T=10s



Phase 1.Volume Fraction
Contour 1
1.000e+000
9.000e-001
8.000e-001
7.000e-001
6.000e-001
5.000e-001
4.000e-001
3.000e-001
2.000e-001
1.000e-001
0.000e+000
ANSYS
R16.2
Academic
0
0.450
0.900 (m)
0.225
0.675
Y
X

B) Task 1 Part B involved measuring the leading edge of the fluid flow and plotting it with respect to time. The measurements were taken at .5, 1, 1.5, 2 and 2.5 seconds. This plot was then fitted with a line using the least squares method. This line gave the average velocity, which was then utilized to find a courant number. The time step size utilized was .01s, and the by measuring the mesh, the delta X was found to be .032 M. It is important in time marching solvers to ensure the Courant number is less than 1, to maintain numerical stability.



The formula for the Courant Number *C* is:

$$C = u\frac{\Delta t}{\Delta x}$$

Solving for *C*:

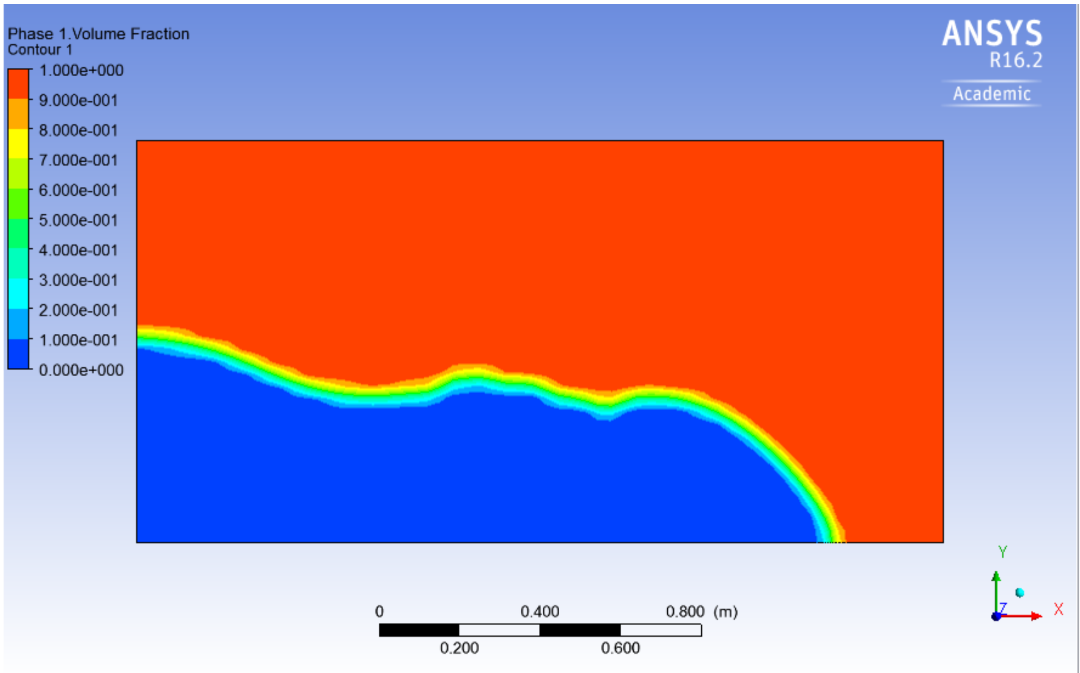
$$C = .323\frac{.01}{.032} = .1009$$

$$C = .1009 < C_{max}$$

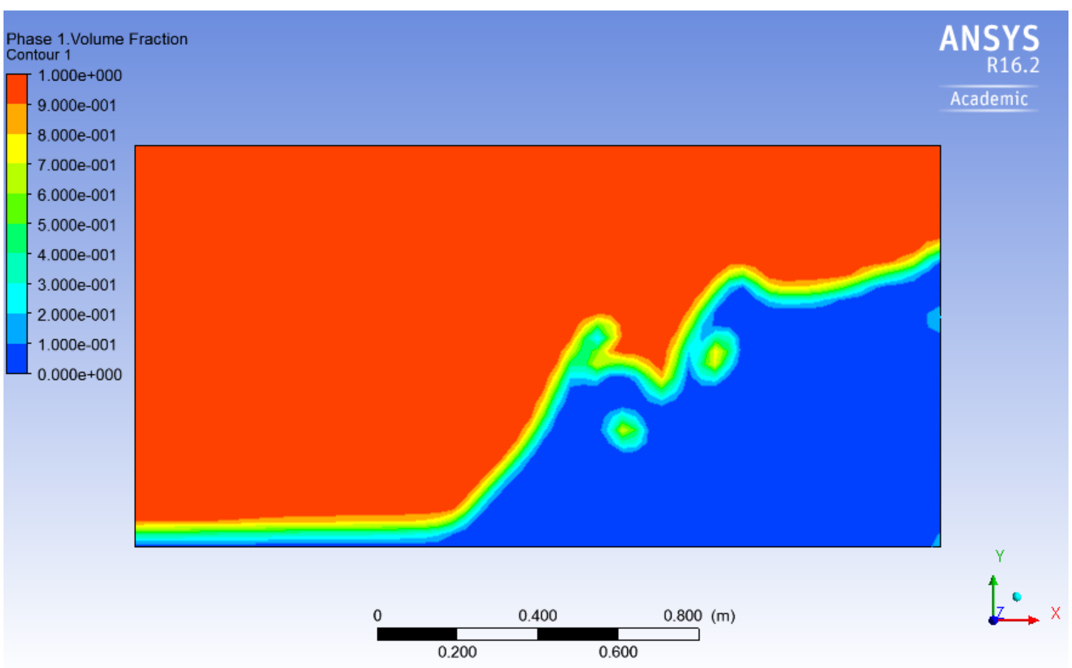
C) Inviscid Model

For the third part of Task 1, the conditions were kept the same, but the model was changed to inviscid rather than K-epsilon. By changing to inviscid, water was effectively treated as having no viscosity. This prevented energy from being bled off during the break, and caused the chaotic splashing and sloshing effects clearly seen on the 6 and 10 second plots. The inviscid model essentially neglects friction, so the flow would continue sloshing back and forth. This is similar to the ball rolling in a semicircle. It will roll up and down the sides of the semicircle forever if there is no friction.

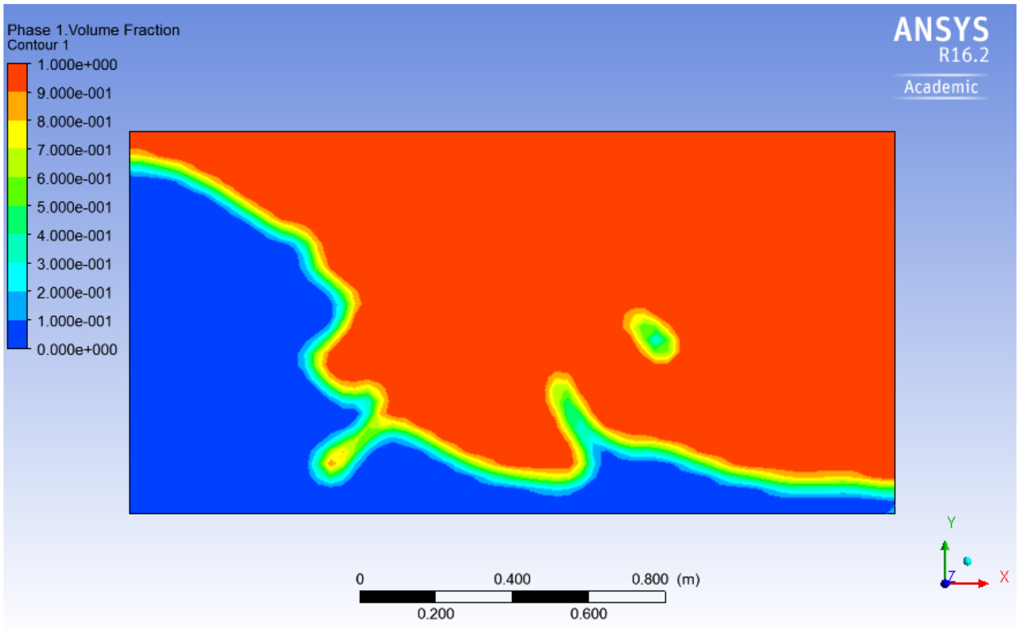
T=2s


Phase 1.Volume Fraction
Contour 1
1.000e+000
9.000e-001
8.000e-001
7.000e-001
6.000e-001
5.000e-001
4.000e-001
3.000e-001
2.000e-001
1.000e-001
0.000e+000
ANSYS
R16.2
Academic
0
0.400
0.800
(m)
0.200
0.600
Y
X

T=6s


Phase 1.Volume Fraction
Contour 1
1.000e+000
9.000e-001
8.000e-001
7.000e-001
6.000e-001
5.000e-001
4.000e-001
3.000e-001
2.000e-001
1.000e-001
0.000e+000
ANSYS
R16.2
Academic
0
0.400
0.800
(m)
0.200
0.600
Y
X

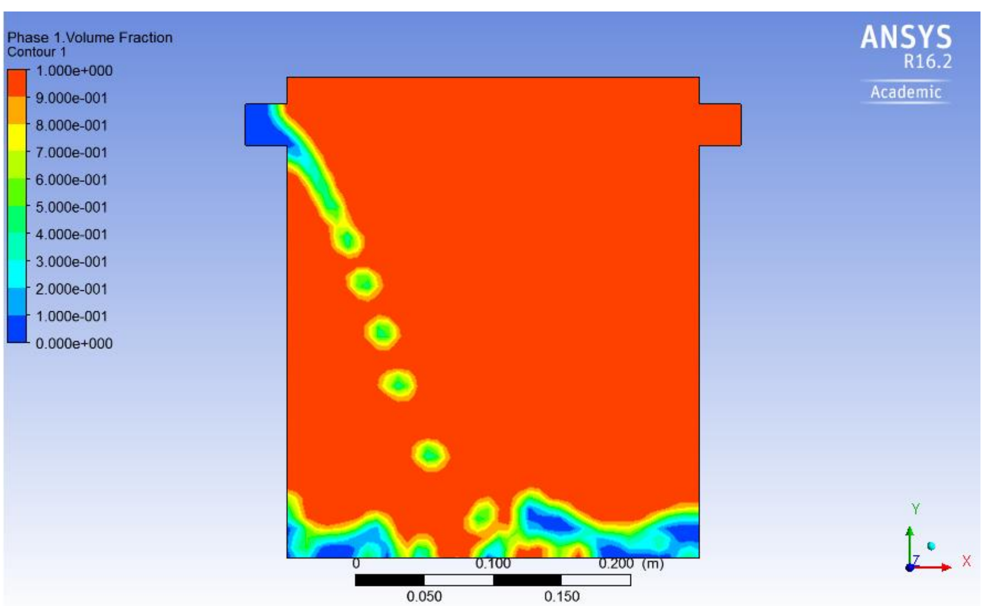
T=10s



Task 2)

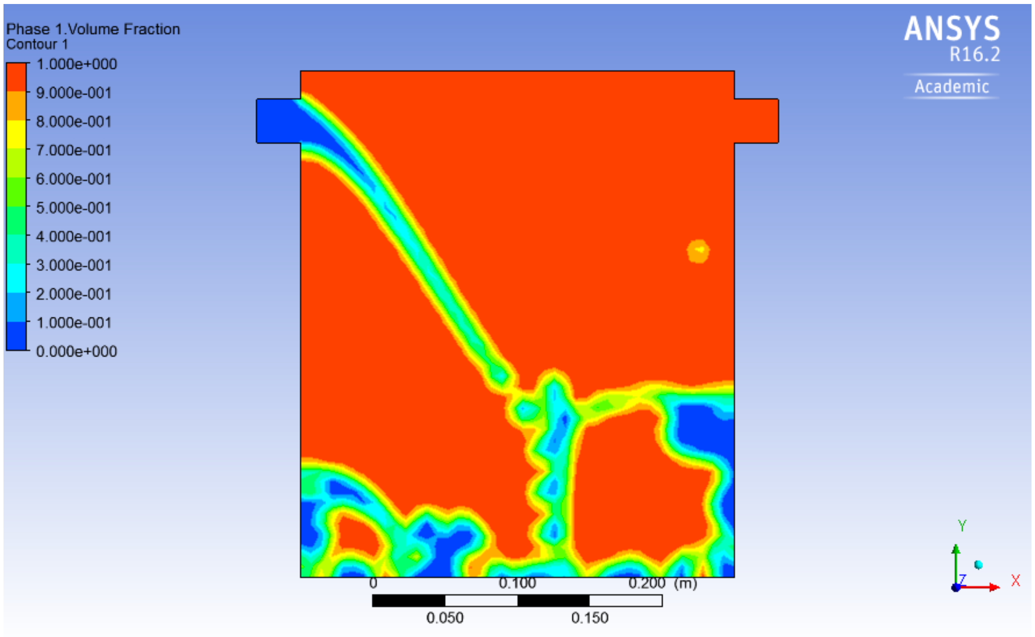
The setup used in task 2 was a turbulence, k-epsilon model with the standard values. The first plot shows the inlet stream having a velocity of .25 M/s. The droplets can be seen, rather than a solid stream, due to the extremely slow flow. This is much similar to the slow dripping of a faucet that is barely opened. The second plot has an inlet velocity of .5 M/s and the stream is much more defined. The water can be seen splashing back in plot 2, forming a swirl of sorts. If the inlet velocity was further increased, a full stream would likely be seen filling the tank with the second phase, water.

T=1s , V=.25 M/s



T=1s

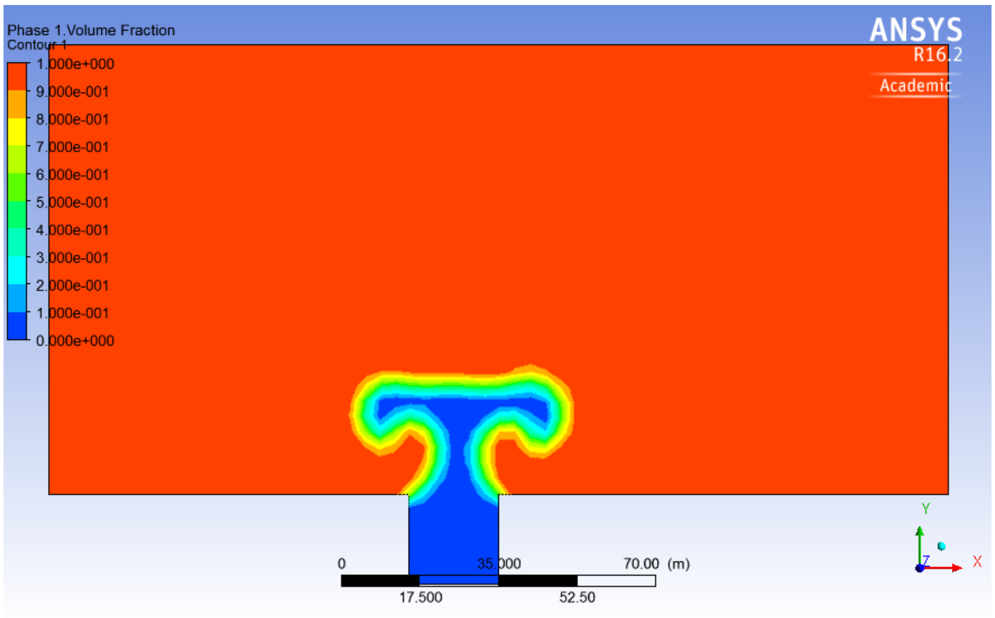
V=.50 M/s



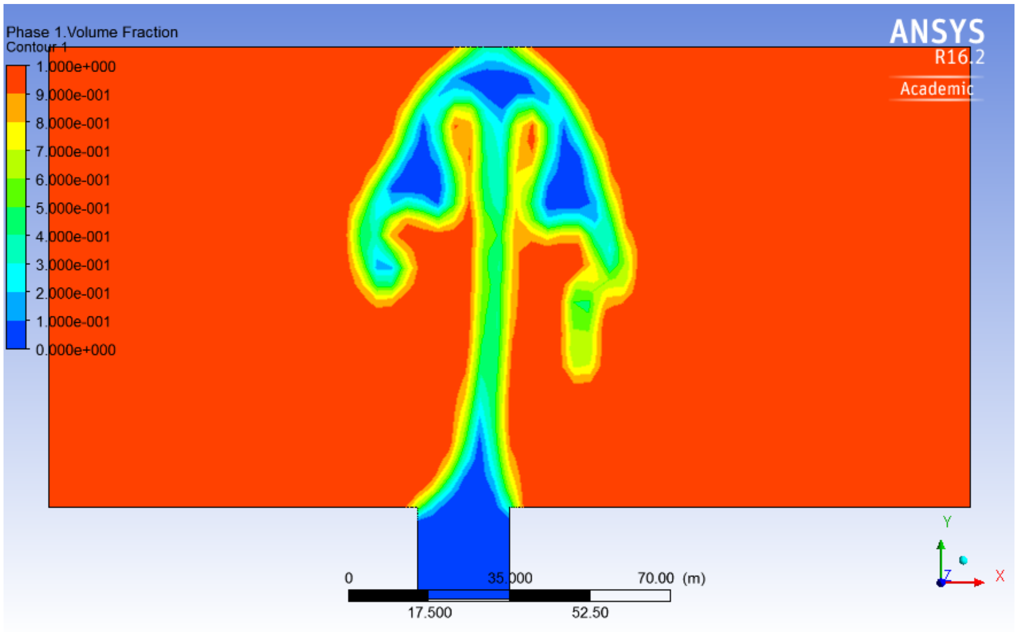
Task 3)

This task simulated a natural gas leak. A methane stream was injected into an open crossflow, to simulate a broken line leaking into moving open air. This simulation utilized a transient 2D, turbulent k-epsilon model similar to the previous tasks. The ambient air speed was set to .1 and 3 M/s for cases A and B respectively. The simulation was ran for 20 seconds of duration, utilizing .01s time steps, with a max of 15 iterations per step. The contour plots of phase volume fraction are shown below.

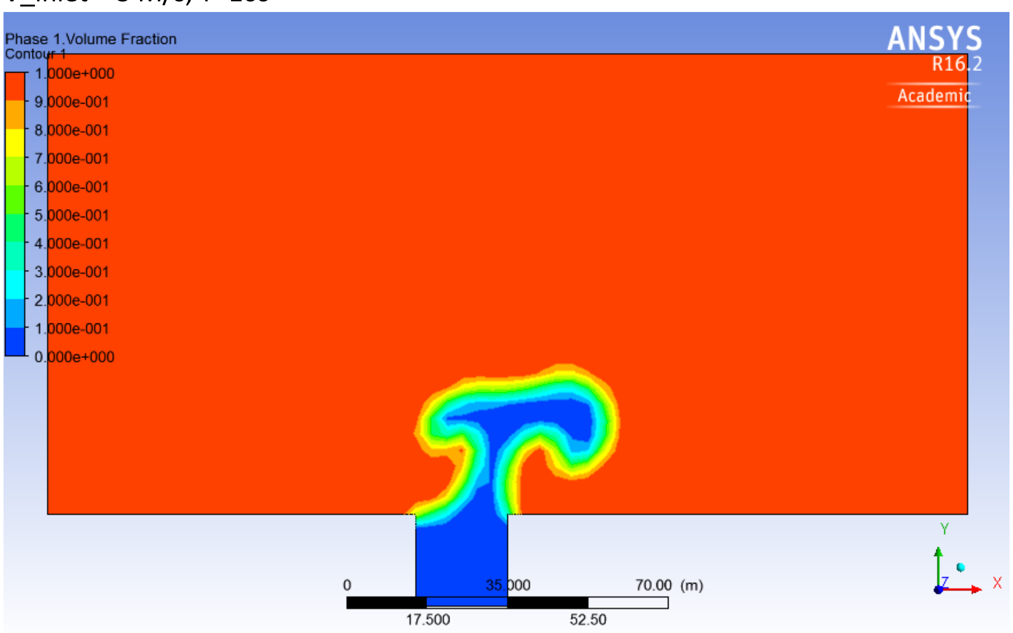
A) V_inlet= .1 M/s, T=10s



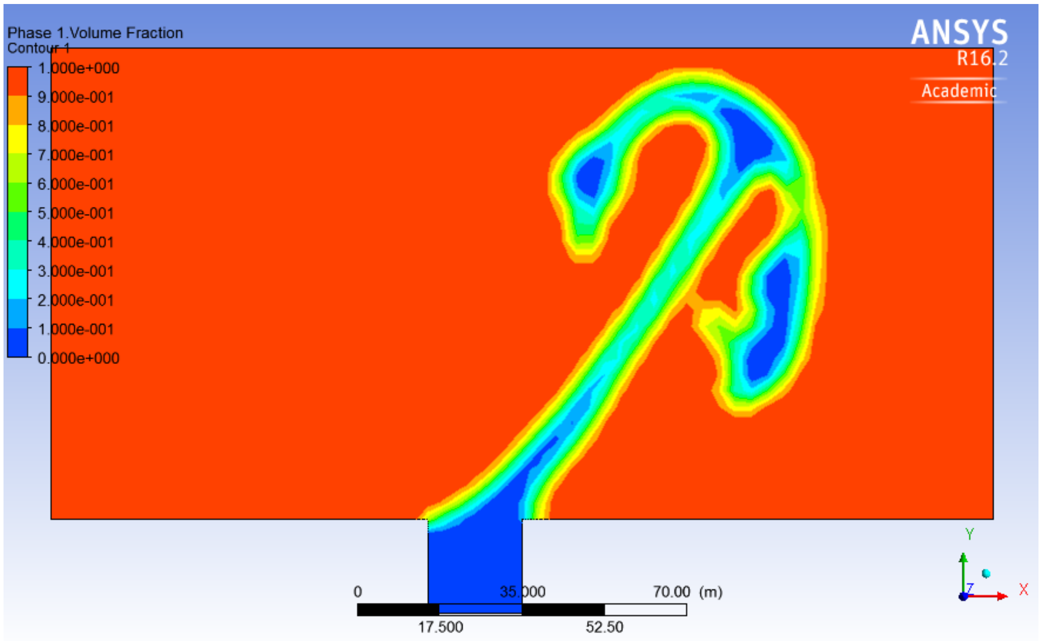
.1 M/s, T=20s



B) V_inlet= 3 M/s, T=10s



3 M/s, T=20s



Phase 1.Volume Fraction
Contour 1
1.000e+000
9.000e-001
8.000e-001
7.000e-001
6.000e-001
5.000e-001
4.000e-001
3.000e-001
2.000e-001
1.000e-001
0.000e+000
ANSYS
R16.2
Academic
0
35.000
70.00 (m)
17.500
52.50
Y
X
Z