

Project # 2

Task 1

Consider a closed rectangular domain with wall at the boundary as illustrated in Fig. 1. Initially (at $t = 0$), the chamber is filled with *liquid kerosene* except that a square subdomain in the lower left corner is filled with *engine oil*. The density and viscosity (both set to constant) of each phase of fluid can be copied from the existing database in Fluent.

(a) Use *viscous-laminar* model to perform a transient simulation and make the contour plots of the *volume fraction* of *engine oil* (if engine oil is defined as phase-2, this would be the volume fraction of phase-2) for the solution at $t = 1$ s, 5 s, and 10 s.

(b) Repeat the task in part (a) but run the simulation using the *inviscid* model instead.

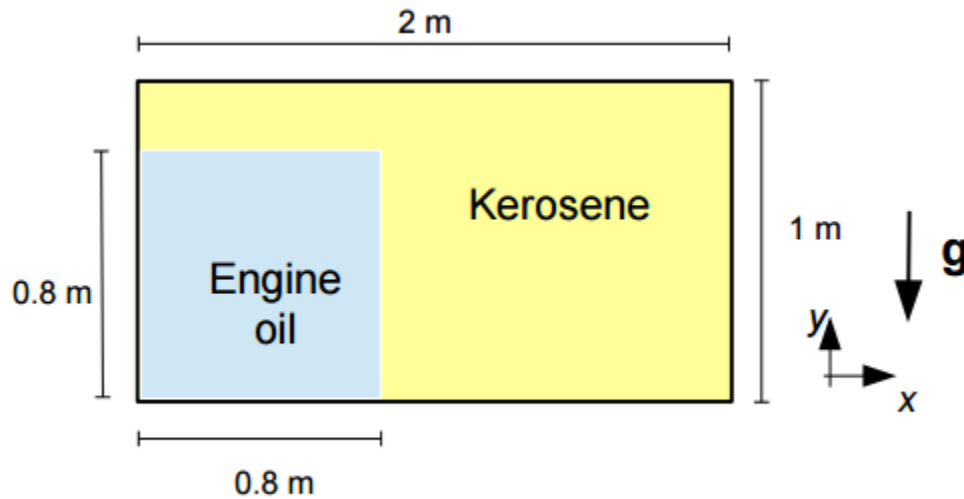


Fig. 1 The geometry and initial distribution of the phases for Task 1.

(c) For the system considered in parts (a) and (b), the total potential energy, PE , and total kinetic energy, KE , are defined by

$$PE = \iint \rho g y dx dy \quad \text{Eq. (1)}$$

$$KE = \iint \frac{1}{2} \rho (u^2 + v^2) dx dy \quad \text{Eq. (2)}$$

where ρ is density and the double integral is over the entire rectangular domain. We define the "*baseline potential energy*", PE_0 , as the value of PE for the terminal state (shown in Fig. 1A), i.e., the solution for the viscous case as $t \rightarrow \infty$ when the system would not move further. In other words, PE_0 is the portion of PE that is not available for conversion to KE . As such, we define the "*available potential energy*" as

$$APE \equiv PE - PE_0 \quad \text{Eq. (3)}$$

As the system evolves in time, we expect exchanges between APE and KE . For the viscous case, upon reaching the terminal state, both APE and KE will approach zero. From the simulations in part (a) and (b), plot APE , KE , and $APE+KE$ as a function of time (please collect the three curves in one plot) for the two cases using *viscous-laminar* and *inviscid* model. Each curve in the plot

should cover at least the range of $0 \leq t < 25$ s and have a temporal resolution of at least 0.1 s. (In other words, each curve will consist of at least 250 data points.) Based on the plots, discuss the effect of viscosity on the behavior of the solution.

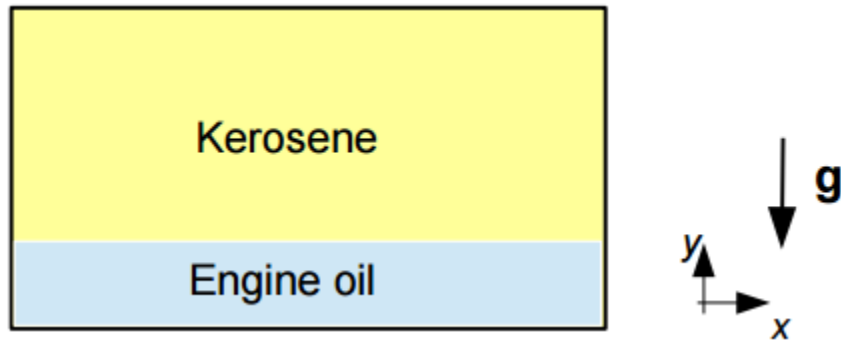
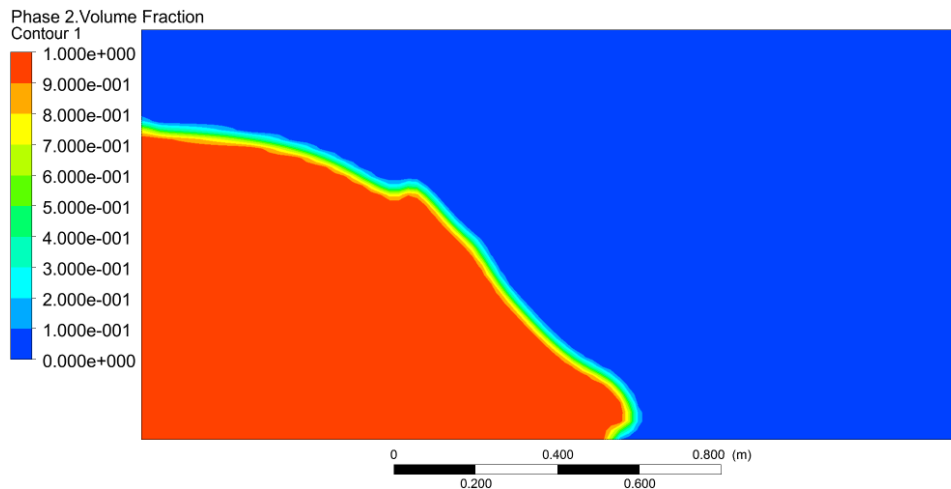


Fig. 1A The terminal state for the viscous case. The potential energy of this state is the “baseline potential energy”, PE_0 .

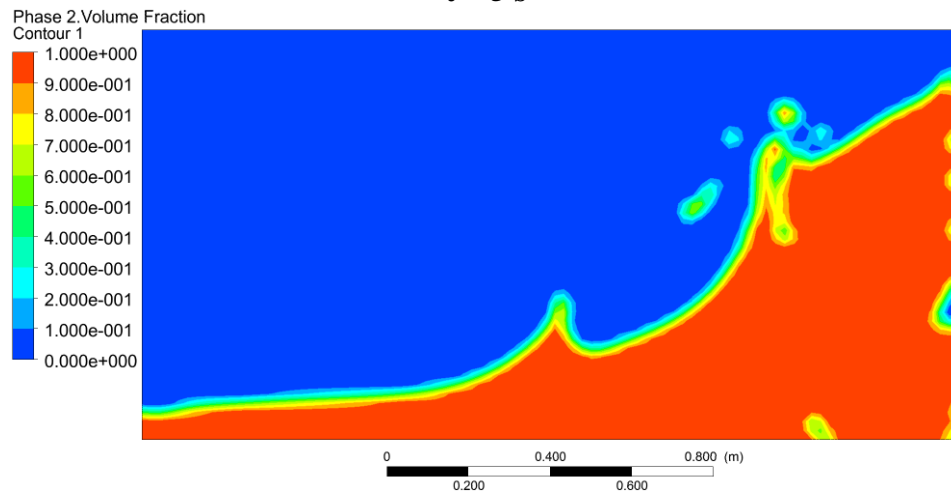
Solution (a):

Here are the contour plots of the volume fraction of engine oil at $t = 1$ s, 5 s, and 10 s using the viscous-laminar model:

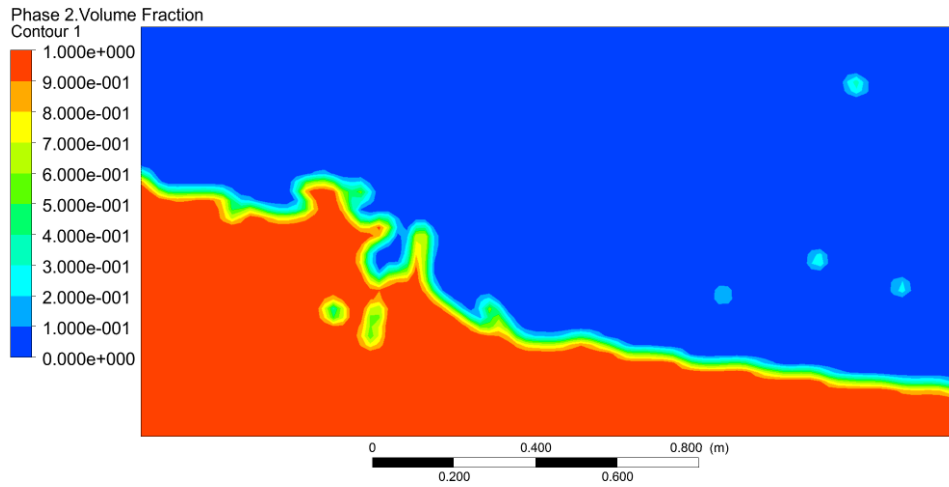
t = 1 s



t = 5 s



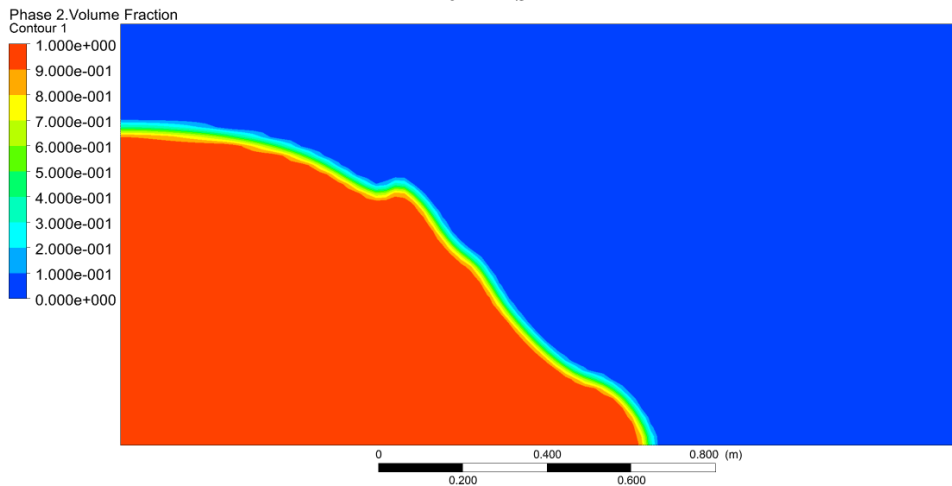
t = 10 s



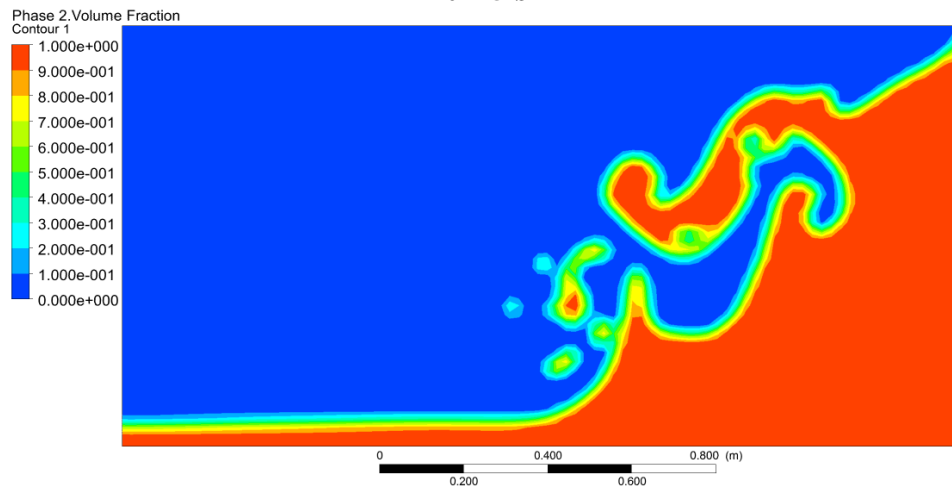
Solution (b):

Here are the contour plots of the volume fraction of engine oil at $t = 1$ s, 5 s, and 10 s using the inviscid model:

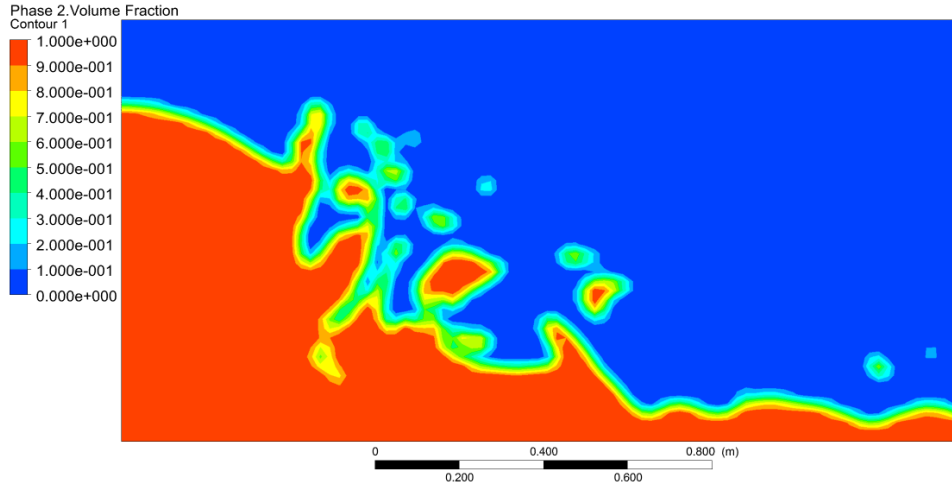
t = 1 s



t = 5 s



t = 10 s



Solution (c):

Before the plots could be made, the baseline potential energy needed to be calculated. Looking at the given figure, the height of the engine oil at the terminal state needed to be found. The initial area of engine oil that is in the initial state is:

$$A_e = (0.8 \text{ m})(0.8 \text{ m}) = 0.64 \text{ m}^2$$

This area will remain conserved throughout the entire problem. This means the area of the engine oil in the terminal state is also 0.64 m^2 . Since the length of the domain is given to be 2 m , the height of the engine oil can be found by dividing the area of the engine oil by 2 m , which gives:

$$h_e = \frac{0.64 \text{ m}^2}{2 \text{ m}} = 0.32 \text{ m}$$

Since the domain is separated into two different fluids at the terminal state, Eq. (1) can be applied to each fluid. The density for each integral will be the density of the fluid and the height will be based off of the height of the engine oil just found. This means the bounds for the y -value will be 0 to 0.32 m for the engine oil and 0.32 m to 1 m for the kerosene. Applying this to both fluids gives:

$$PE_e = \int_0^{0.32} \int_0^2 \rho_e g y \, dx \, dy = \int_0^{0.32} 2 \rho_e g y \, dy = (\rho_e g y^2) \Big|_{y=0}^{y=0.32} = (0.1024 \text{ m}^3) \rho_e g$$

$$PE_k = \int_{0.32}^1 \int_0^2 \rho_k g y \, dx \, dy = \int_{0.32}^1 2 \rho_k g y \, dy = (\rho_k g y^2) \Big|_{y=0.32}^{y=1} = (0.8976 \text{ m}^3) \rho_k g$$

The baseline potential energy will be the sum of these two potential energy values, which is

$$PE_0 = PE_e + PE_k = 0.1024 \rho_e g + 0.8976 \rho_k g$$

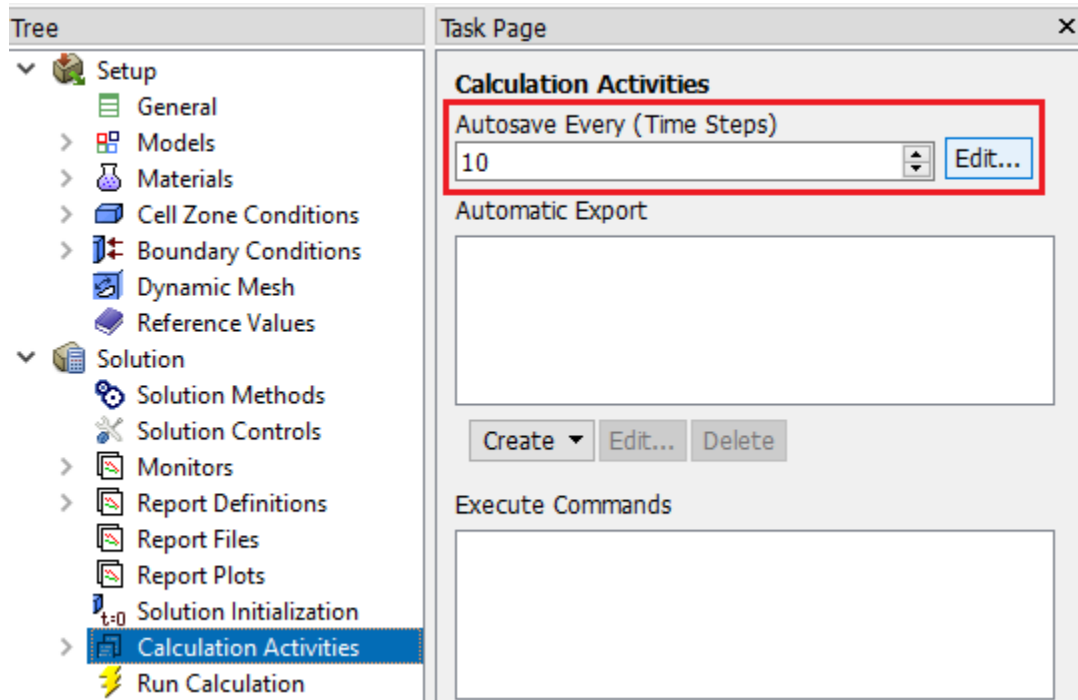
Substituting in the densities and the value for gravity at earth's surface gives

$$PE_0 = \left(9.8066502 \frac{\text{m}}{\text{s}^2}\right) \left[(0.1024 \text{ m}^3) \left(889 \frac{\text{kg}}{\text{m}^3}\right) + (0.8976 \text{ m}^3) \left(780 \frac{\text{kg}}{\text{m}^3}\right) \right]$$

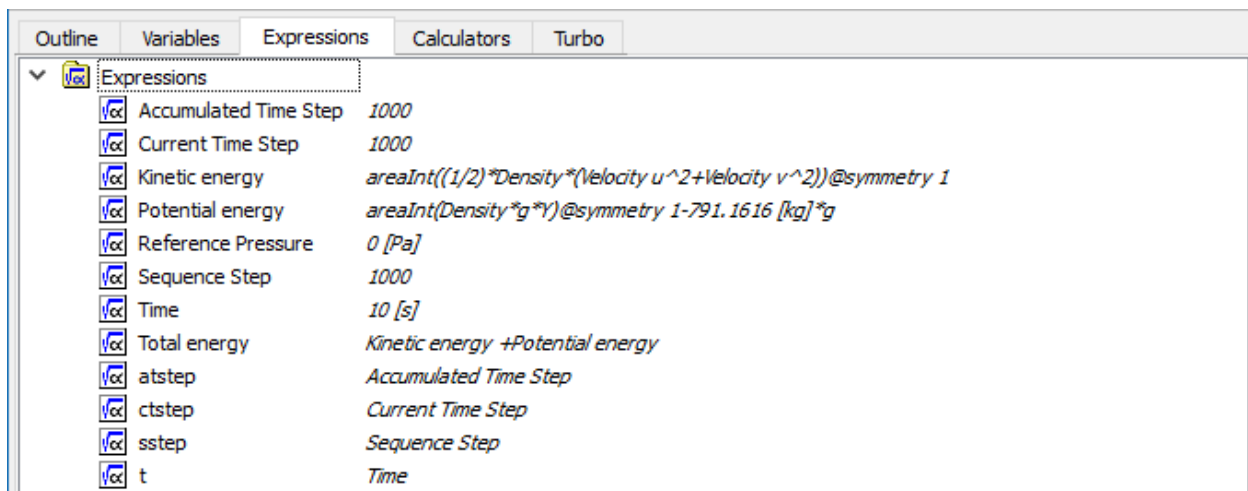
$$\boxed{PE_0 = 7758.6451 \text{ N}}$$

This baseline potential energy can be used in Eq. (3) to get the available potential energy. To plot the kinetic, available potential, and total energy, these values had to be extracted during the

simulations. There is an auto-save feature in Ansys Fluent that can be used to save data from a simulation for the time steps that are chosen. An image of this option can be seen below:

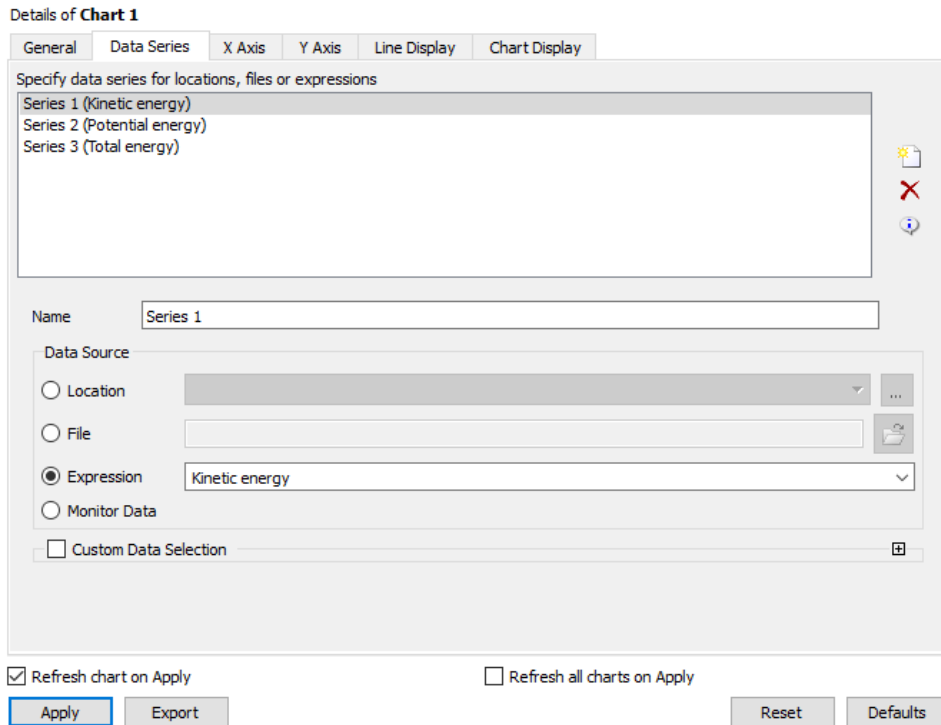


Some of the values that are auto-saved are values that are needed in Eq. (1) and Eq. (2) to find the kinetic and potential energy. These values could be used in CFD-Post to find the kinetic, available potential, and total energy at each time step that was saved. To do this, expressions were made for the kinetic, available potential, and total energy like what was done in Project 1 and what was done in Homework 1 by William Burk¹. Here are the expressions:

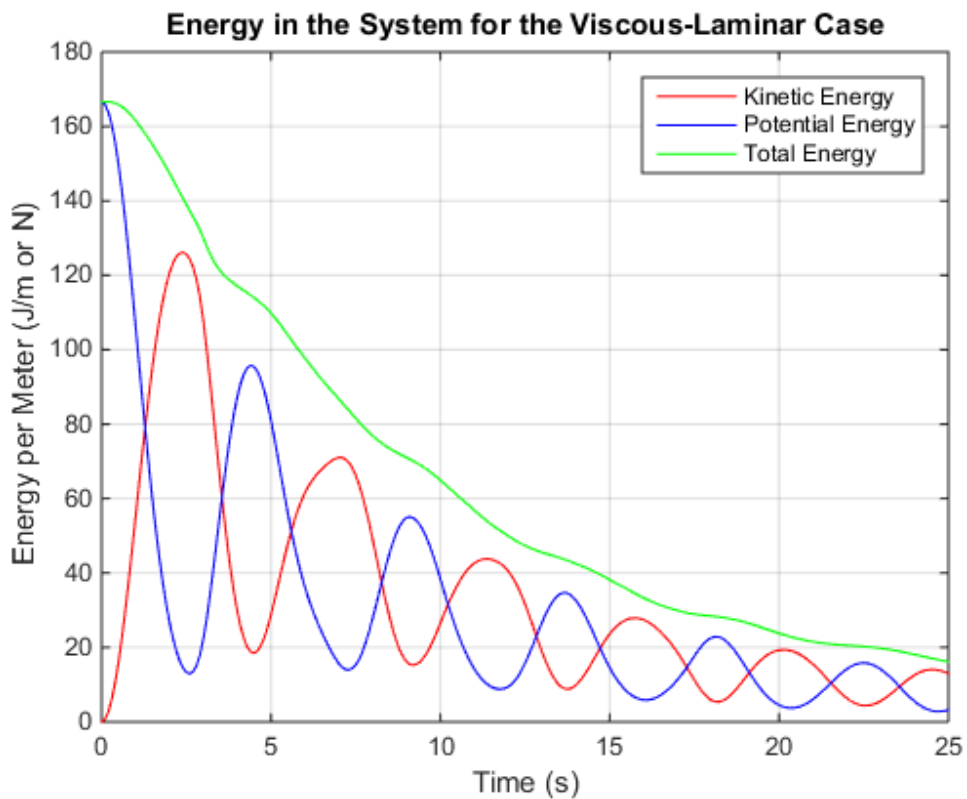


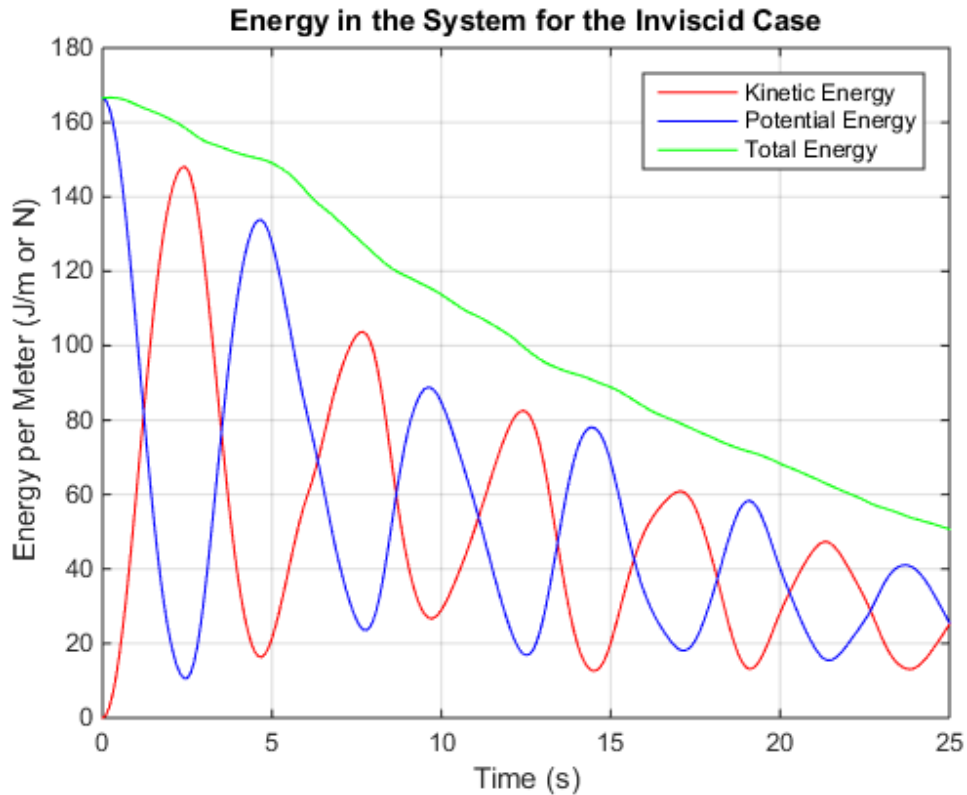
The next step was to create a chart. The transient option was chosen in the “General” tab. In the “Data Series” tab, series of data could be created from a chosen file, an expression, or data from a monitor. The expression option was chosen and each of the expressions for energy were used to create a series. Since the required values for these expressions were already auto-saved, each series contained the energy values at all of the auto-saved time steps. These series were then

exported to MATLAB to be plotted even though the “Line Display” tab could have been used to plot the data. Here is what the chart options look like:



Here are the plots of the kinetic, available potential, and total energy for the viscous-laminar and inviscid cases from the exported data:





As can be seen in the plots, the viscous-laminar case ends up losing energy much faster than the inviscid case. This makes sense because the inviscid case does not have viscous effects while the viscous-laminar case does. As discussed in class, viscous effects cause momentum from one fluid “parcel” to be transferred normal to the direction of motion to another slower moving fluid “parcel”. This means momentum is lost in the main direction of motion, which means the velocity has been lowered. This decrease in the velocity means the kinetic energy is lowered as well. Since the available potential energy can be transferred to kinetic energy, the available potential energy can also be lowered so the total energy ends up decreasing rapidly.

Task 2

A 2-D chamber, illustrated in Fig. 2, consists of a main chamber and two symmetric side pipes for the inlet and outlet. All boundaries are solid walls except the *velocity inlet* at left and *pressure outlet* at right. Initially, the entire chamber including the side pipes is filled with *air*. At $t > 0$, *water* is injected through the inlet which produces a jet that gradually fills the chamber with water (as air is simultaneously pushed out of the chamber through the outlet). Define air as *phase-1* (the *primary phase*) and water *phase-2*, the key setups at the boundaries are:

- (i) *Volume fraction* = 1 for *phase-2* at the *velocity inlet*
- (ii) *Velocity* = a given V_{inlet} (see below) for *mixture* at the *velocity inlet*
- (iii) *Gauge pressure* = 0 for *mixture* at both inlet and outlet (this is the default anyway)

Using viscous (turbulence) *k-epsilon model*, perform transient simulations for the two cases:

- (a) $V_{inlet} = 0.3$ m/s. For this case, make the contour plot of the *volume fraction* of *phase-2* (i.e., *water*) for the solution at $t = 2$ s, 4 s, 6 s.

(b) $V_{inlet} = 0.6$ m/s. For this case, make the contour plot of the *volume fraction of phase-2* (i.e., *water*) for the solution at $t = 1$ s, 2 s, 3 s.

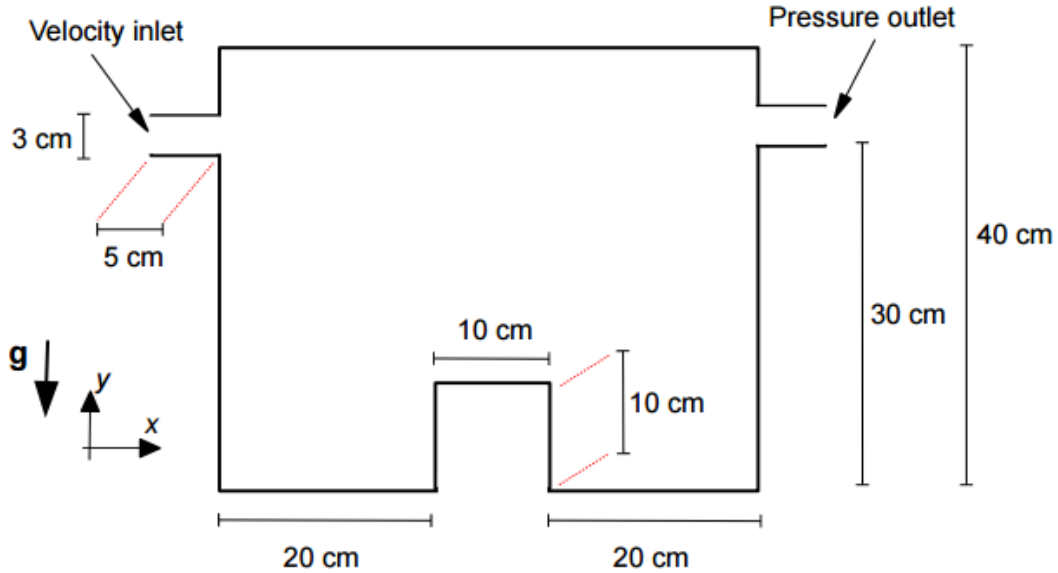
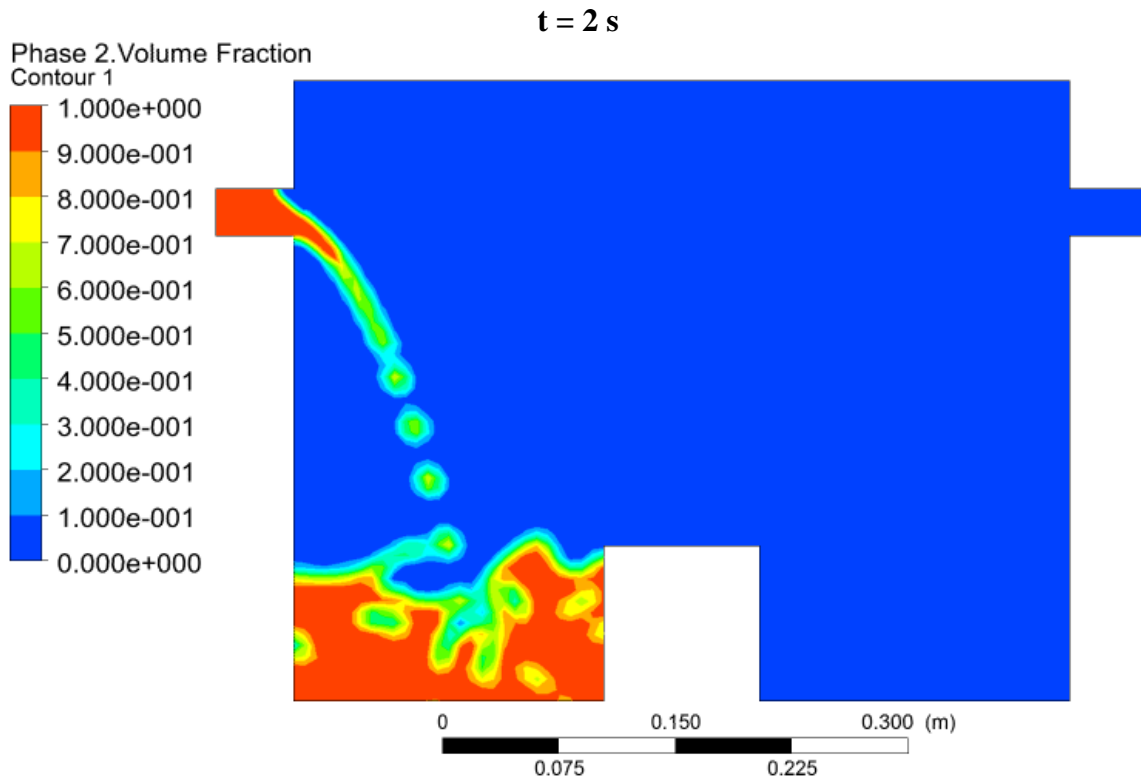


Fig. 3 The geometry of the system for Task 2.

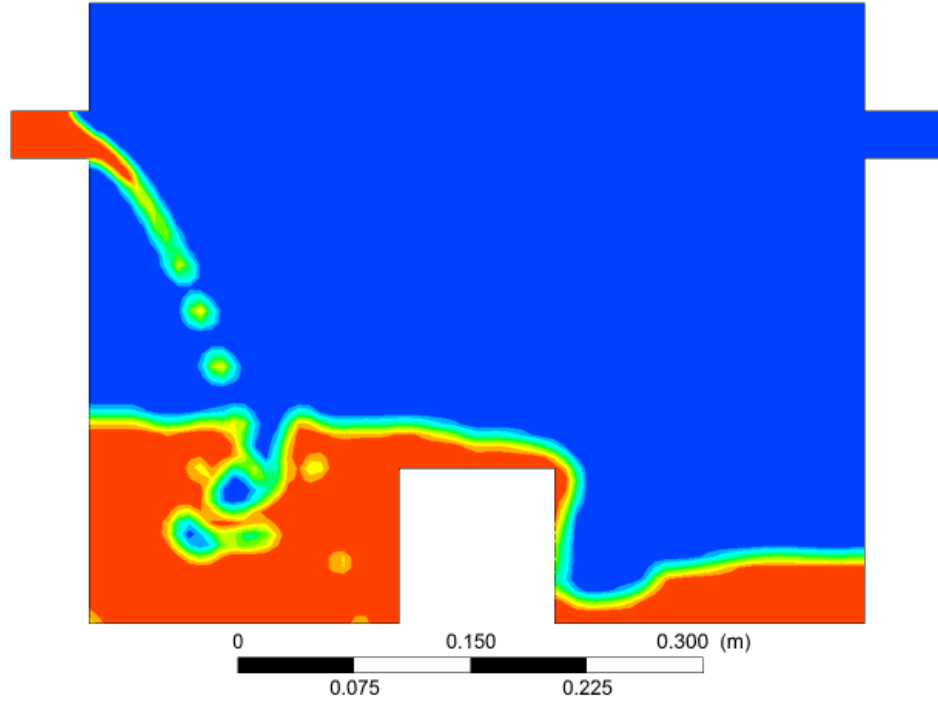
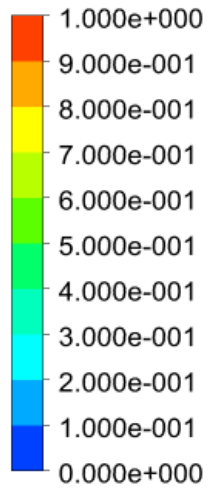
Solution (a):

Here are the contour plots of the volume fraction of water at $t = 2$ s, 4 s, and 6 s when the inlet velocity is set to 0.3 m/s:



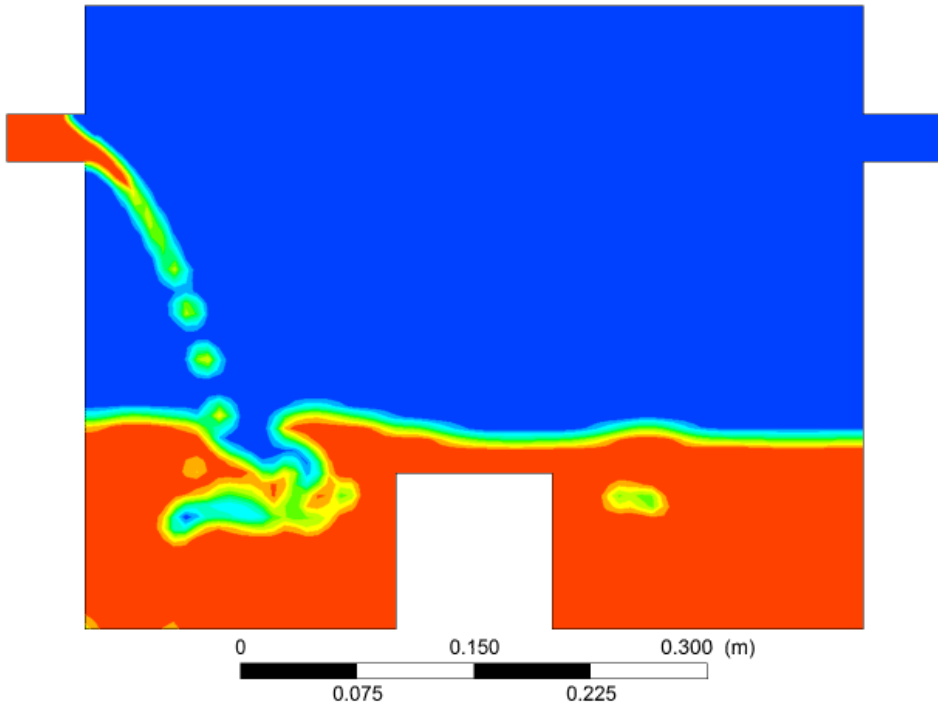
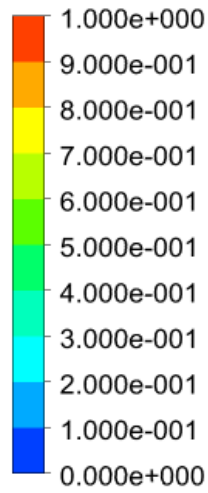
t = 4 s

Phase 2. Volume Fraction
Contour 1



t = 6 s

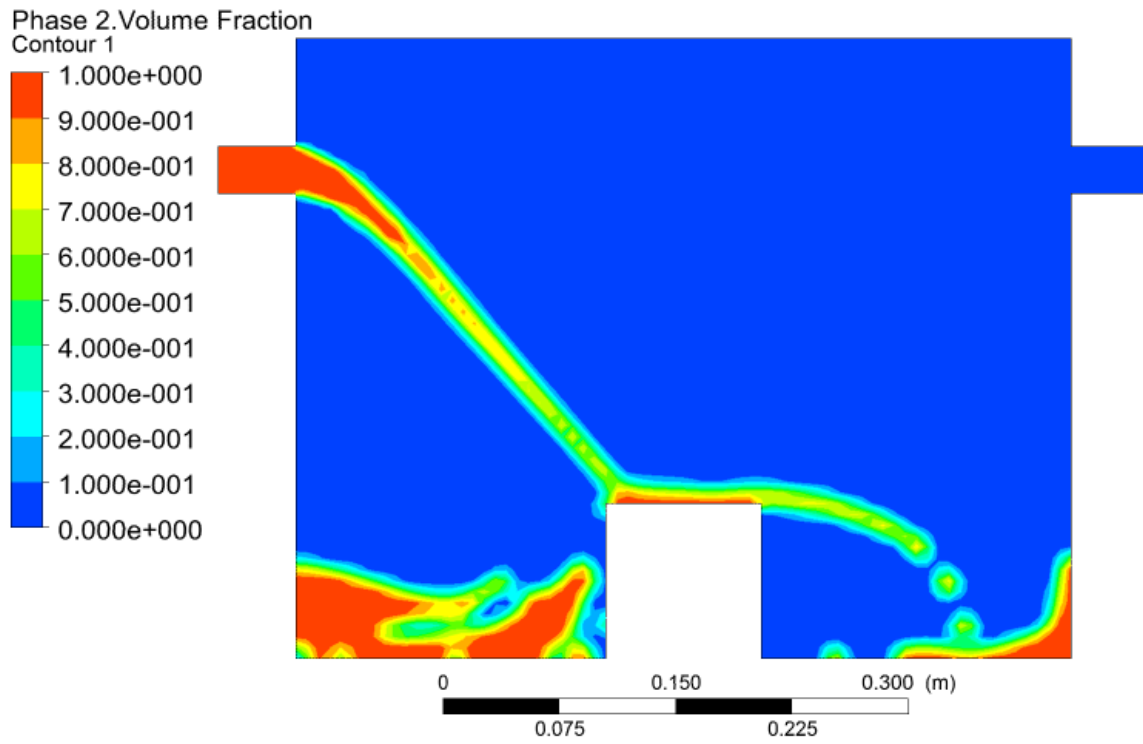
Phase 2. Volume Fraction
Contour 1



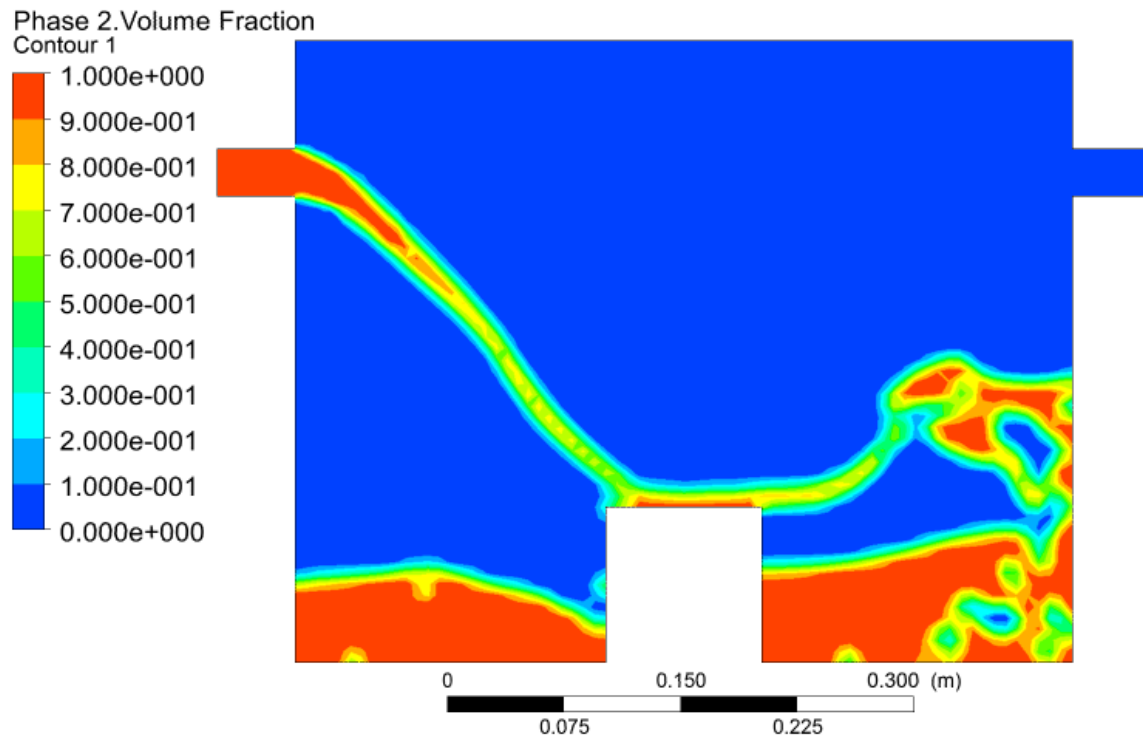
Solution (b):

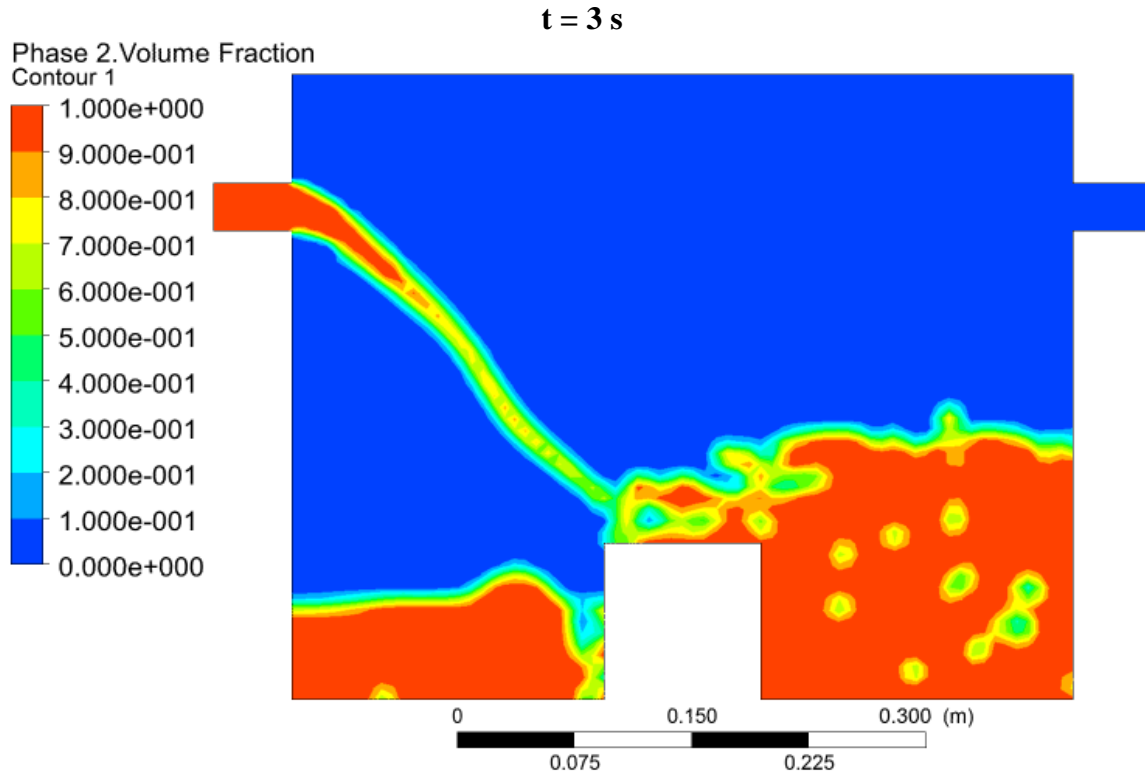
Here are the contour plots of the volume fraction of water at $t = 1$ s, 2 s, and 3 s when the inlet velocity is set to 0.6 m/s:

t = 1 s



t = 2 s





Task 3

This case emulates the situation when natural gas is released into open air from an underground reservoir. We nominally choose *methane* (CH_4) to represent natural gas. The density and viscosity (both set to constant) of methane can be copied from the existing database in Fluent.

The 2-D domain for the simulation is shown in Fig. 3. The bottom of the domain is wall except the opening at *velocity-inlet-1*, through which methane is injected into the domain. The whole left boundary is *velocity-inlet-2* through which normal air will be blown in. The whole top boundary is *outlet-1*, and right boundary *outlet-2*. Define air as *phase-1* (the primary phase) and methane as *phase-2*, the key setups at the boundaries are:

- (i) *Volume fraction* = 1 for *phase-2* at *velocity-inlet-1*
- (ii) *Velocity* = 5 m/s for *mixture* at *velocity-inlet-1*
- (iii) *Volume fraction* = 0 for *phase-2* at *velocity-inlet-2*
- (iv) *Velocity* = a given V_{inlet} (see below) for *mixture* at *velocity-inlet-2*
- (v) *Gauge pressure* = 0 for *mixture* at all inlets. (This is the default)
- (vi) The boundary conditions at both *outlet-1* and *outlet-2* are set to *outflow*.

Using viscous (turbulence) *k-epsilon model*, run two simulations with (a) $V_{\text{inlet}} = 0.2 \text{ m/s}$, and (b) $V_{\text{inlet}} = 2 \text{ m/s}$ for *velocity-inlet-2*. For both cases, the inlet velocity at *velocity-inlet-1* (at bottom) is fixed at 5 m/s. For each case, make the contour plots of the *volume fraction* of *phase-2* for the solution at $t = 5 \text{ s}$ and 10 s . (Two plots for each case.)

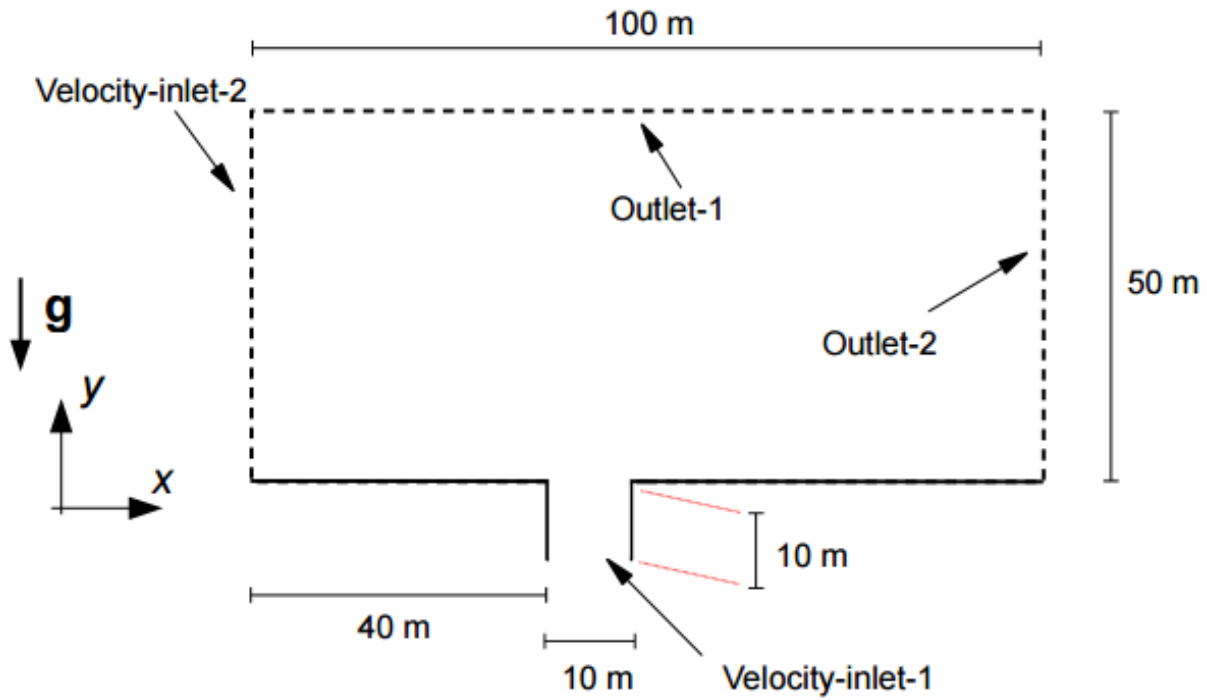
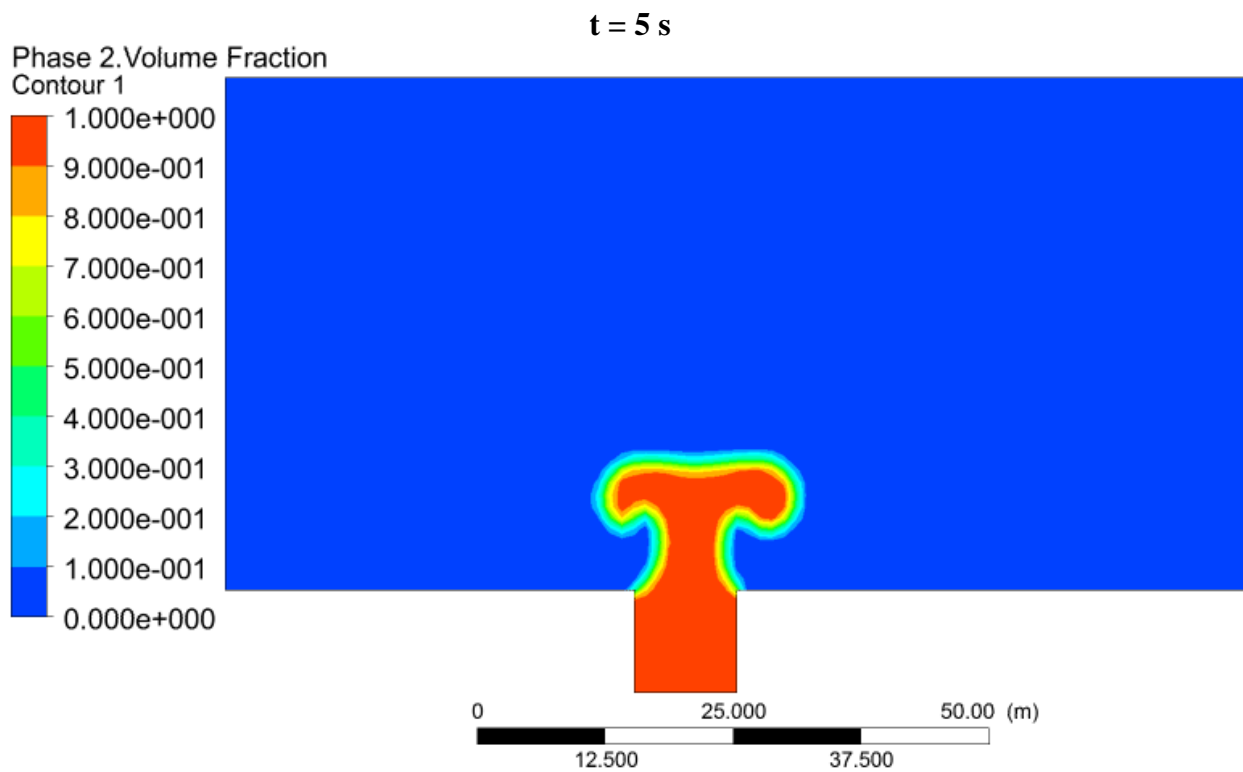


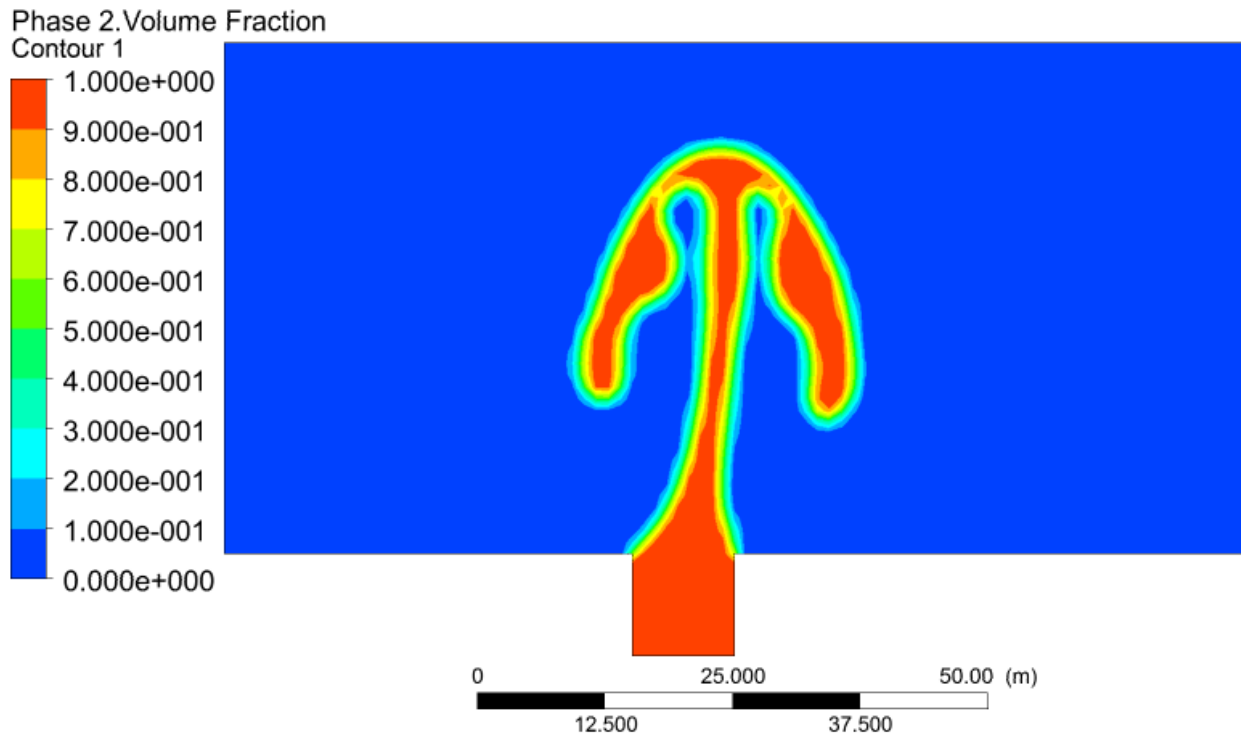
Fig. 3 The geometry of the system for Task 3.

Solution (a):

Here are the contour plots of the volume fraction of methane at $t = 5$ s and 10 s when in inlet velocity for *velocity-inlet-2* is set to 0.2 m/s:



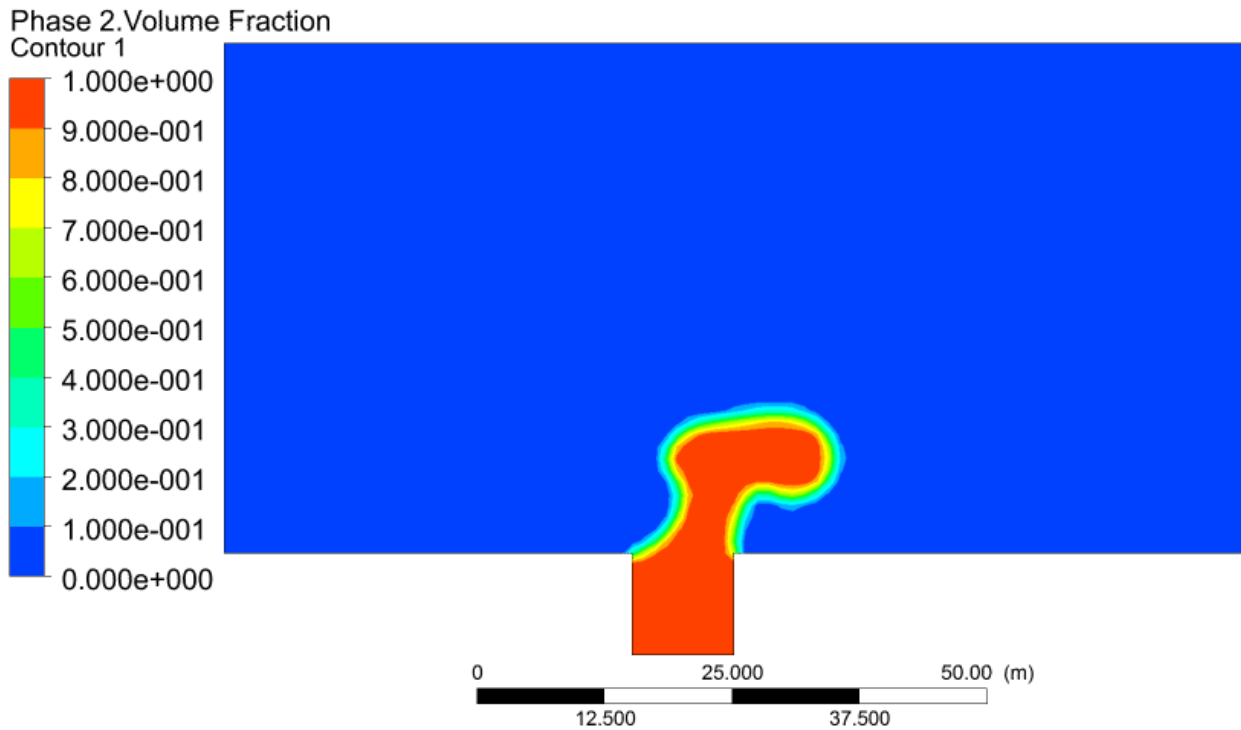
t = 10 s

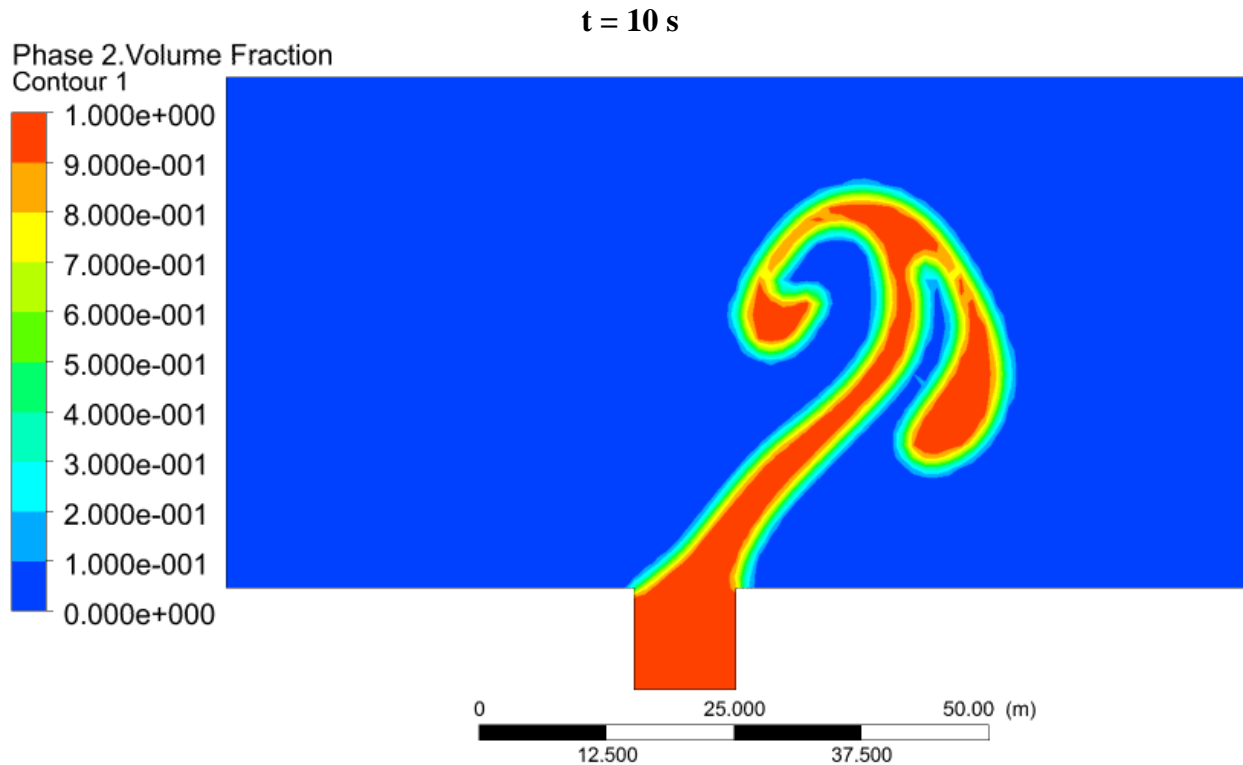


Solution (b):

Here are the contour plots of the volume fraction of methane at $t = 5$ s and 10 s when in inlet velocity for *velocity-inlet-2* is set to 2 m/s:

t = 5 s





Task 4

(a) Consider a 2-D system with an inclined plate that forms a 15° angle with the ground as shown in Fig. 4. At $t = 0$, a drop of engine oil is placed on the plate and it is shaped like a semicircle with a radius of 1 cm. Use Ansys-Fluent to simulate the evolution of this drop and show the shape of the blob of engine oil at $t = 0, 0.1 \text{ s}, 1 \text{ s},$ and 10 s .

(b) Consider the 3-D version of the system in Part (a) with the initial drop of engine oil shaped like a *hemisphere* with a radius of 1 cm. Use Ansys-Fluent to simulate the evolution of this drop and show the shape of the blob of engine oil at $t = 0, 0.1 \text{ s}, 1 \text{ s},$ and 10 s . (It is part of your job to find a way to present the 3-D structure of the blob. A suggestion is to show the iso-surface of $\text{VF} = 0.9$ where VF is the volume fraction of engine oil.)

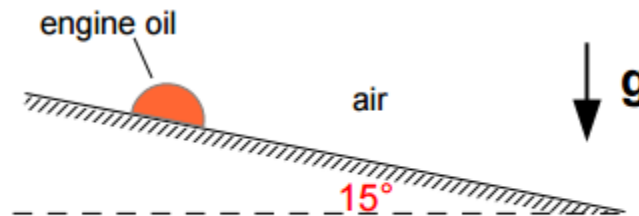


Fig. 4 The initial state of the system considered in Task 4.

Solution (a):

The first step to this problem was setting up the domain. The given domain is a plate at a 15° angle, but this can be changed to make setting up the simulation easier. What was done is instead of having the plate at 15° , the plate was rotated so that it is flat or at an angle of 0° . At the same

time, the vector for gravity was rotated by 15° as well so that it is no longer acting directly downwards and has a component in the positive x-direction as well. Breaking the new gravity vector into horizontal and vertical components gives:

$$g_y = \left(9.80665 \frac{m}{s^2}\right) \cos 15^\circ = 9.47250 \frac{m}{s^2}$$

$$g_x = \left(9.80665 \frac{m}{s^2}\right) \sin 15^\circ = 2.53815 \frac{m}{s^2}$$

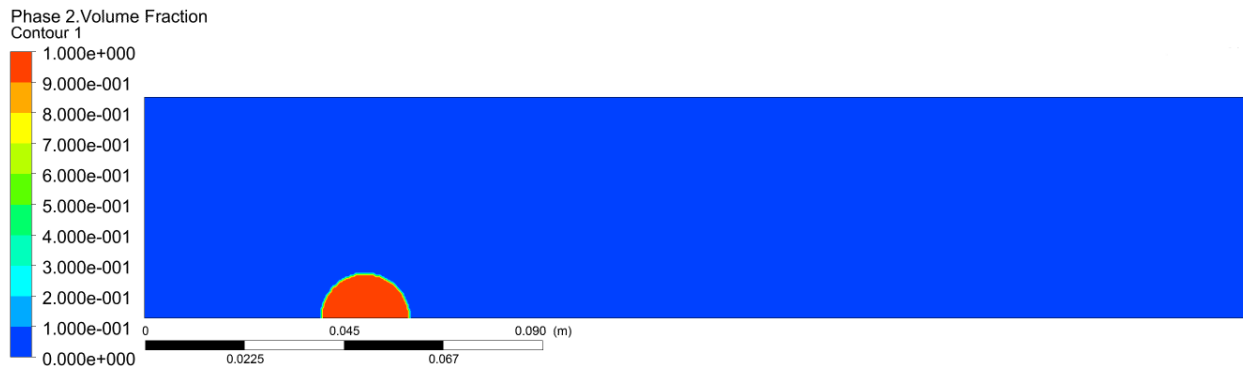
The vertical component is acting downwards of course and was entered as a negative value when put into Fluent.

The next step was to determine the size of the domain to use for the simulation. Since the droplet is only 1 cm tall and won't get any taller the domain does not need to be very tall. The droplet would end up moving some of the air around it as it moves so the height of the domain was set to 5 cm to be safe. The length of the plate was more difficult to determine. If it was assumed that the droplet is a particle, then it was found that the droplet would travel about 127 m in 10 s. This is a ridiculous assumption to make so a much smaller domain length was chosen to be used. To determine an accurate domain length to use, a simulation of this 2-D case was performed using a domain length of 10 m. At the end of this simulation, it was found that the droplet moved only about 10 cm during that 10 s. To be safe, the domain length was chosen based off of this to be 20 cm past the center of the droplet at $t = 0$ s and 5 cm before the center of the droplet. This made the dimensions of the domain **5 cm tall** and **25 cm long**, with 5 cm being before the droplet.

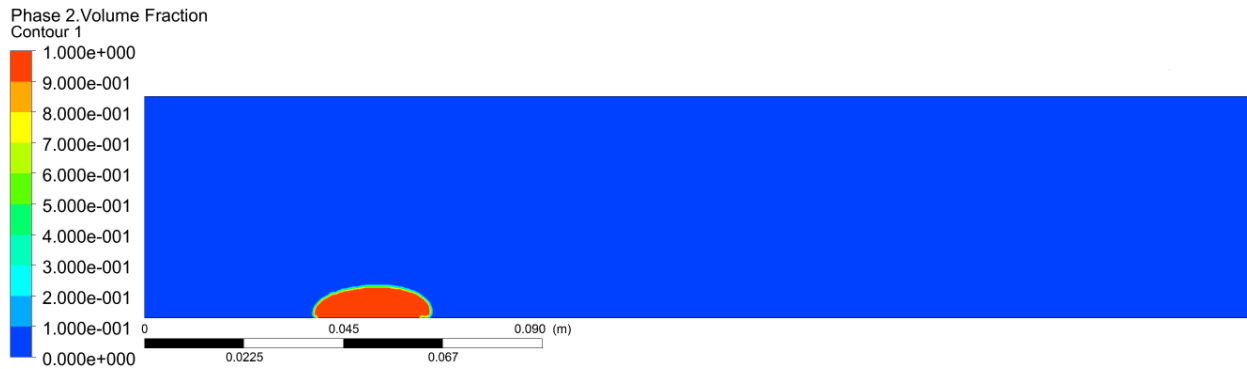
The next step was to determine the boundary conditions to use for the domain. The boundary condition for the plate was of course a wall. Since the other three boundaries of the domain are a decent distance from the droplet and the air that is influenced by its motion, it could be possible to use walls for the other three boundaries. However, to make it more accurate, the left boundary was set as an inlet with no incoming velocity while the top and right boundaries were set as outflows like what was done in Task 3. These boundary conditions would give the best accuracy.

With the domain and boundary conditions known, the simulation was able to be performed. As recommended, the viscous-laminar model was used. Here are the contour plots of the volume fraction of the engine oil at $t = 0$ s, 0.1 s, 1 s, and 10 s. Note that the horizontal component of gravity is pointed to the right:

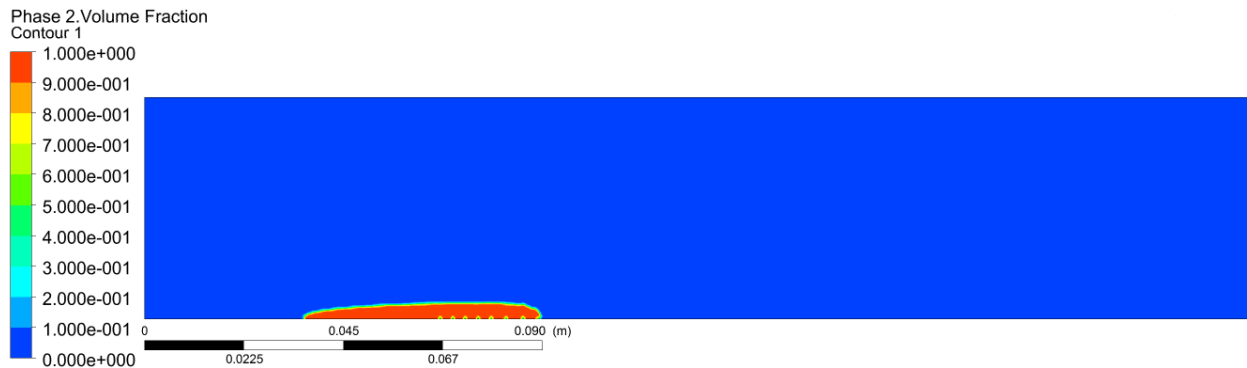
t = 0 s



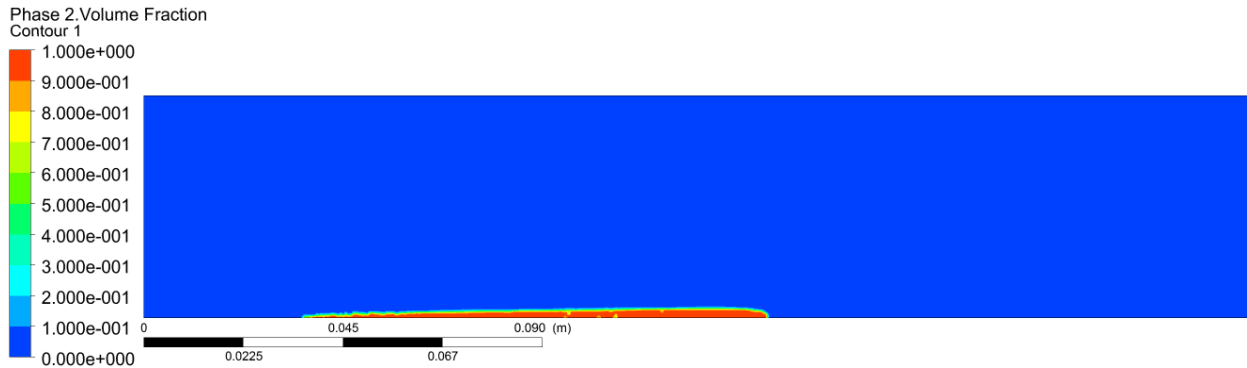
t = 0.1 s



t = 1 s



t = 10 s



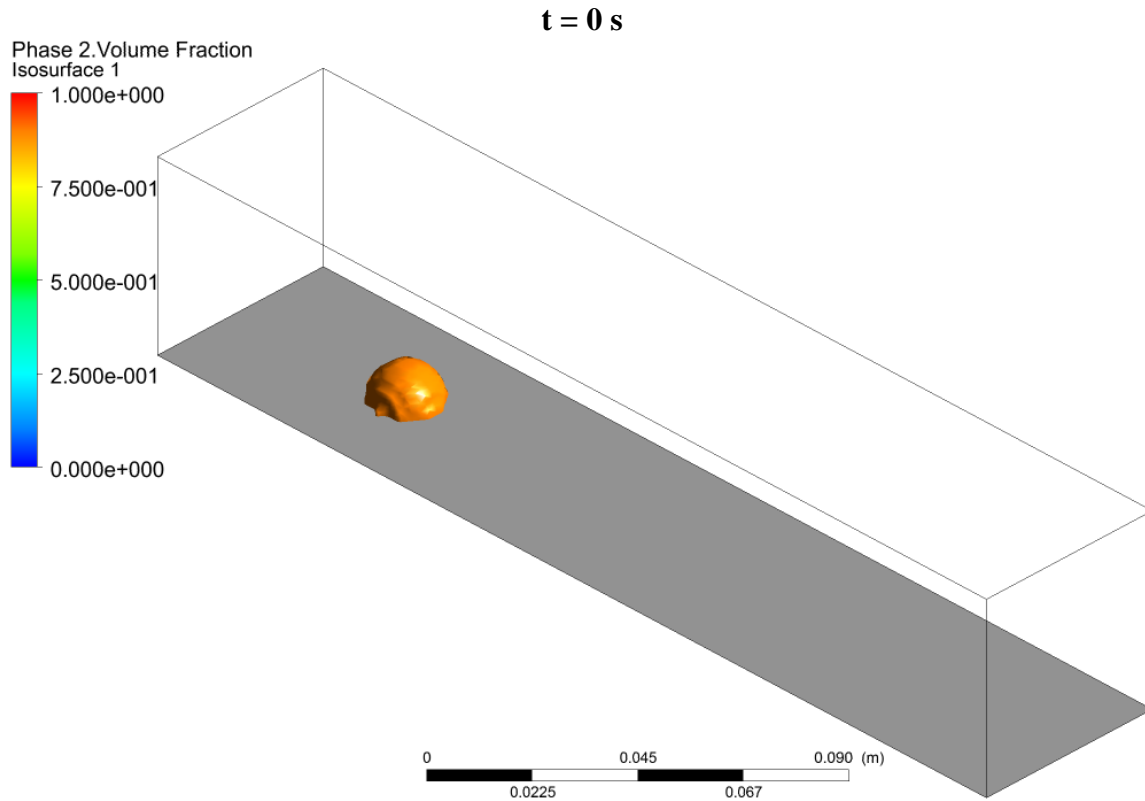
As can be seen above, the droplet quickly flattens out and slowly begins moving to the right.

Solution (b):

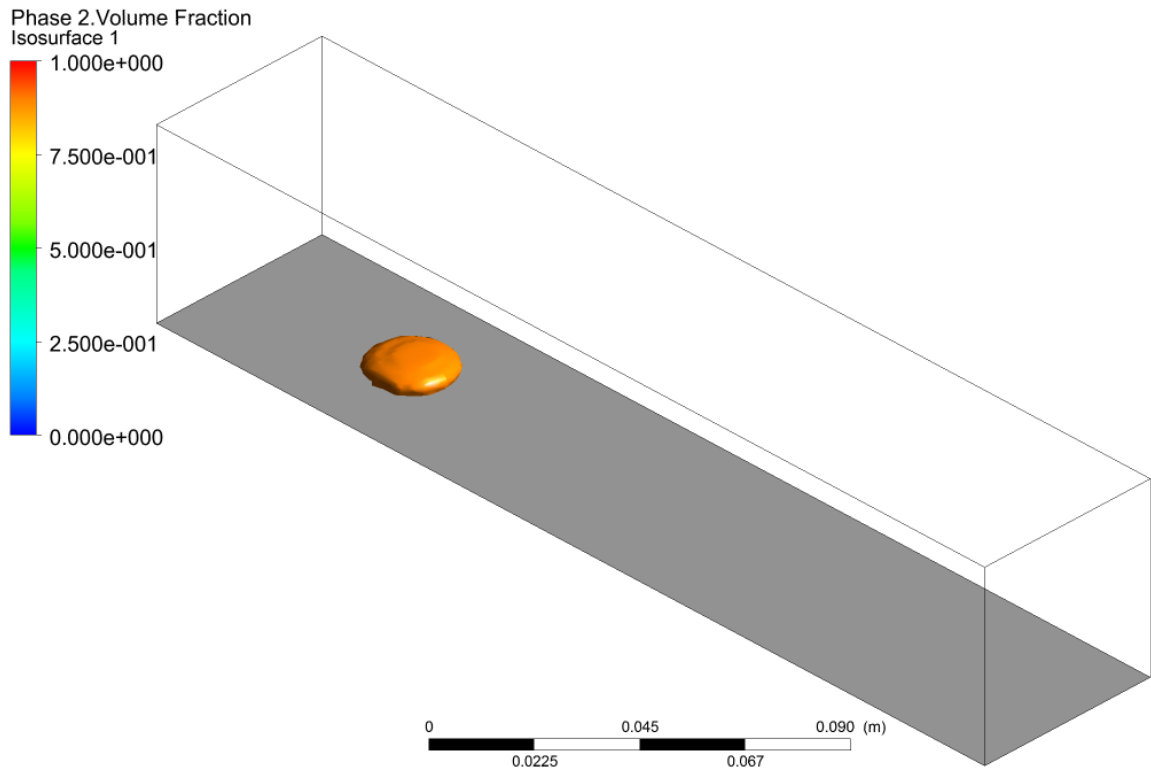
For the 3-D case, the domain and the boundaries were fairly similar to the 2-D case. The new z-component was made normal to the x- and y-directions used in the 2-D case. This means that there was no component of the gravity in the z-direction. The length of the domain in the x-direction and y-direction were kept the same from the 2-D case. Since the droplet shouldn't move much in the z-direction, the length of the domain in the z-direction was set to 2.5 cm on either side of the droplet or 5 cm total. This made the dimensions of the domain **5 cm tall, 5 cm deep** with the droplet in the middle, and **25 cm long** with 5 cm being before the droplet. The boundary conditions were kept the same as the 2-D case with a wall on the bottom, an inlet with no incoming velocity on the left face, and outflows for the top and right face. The two new

boundaries, which would have normal vectors pointing into and out of the screen, were also made into outflows.

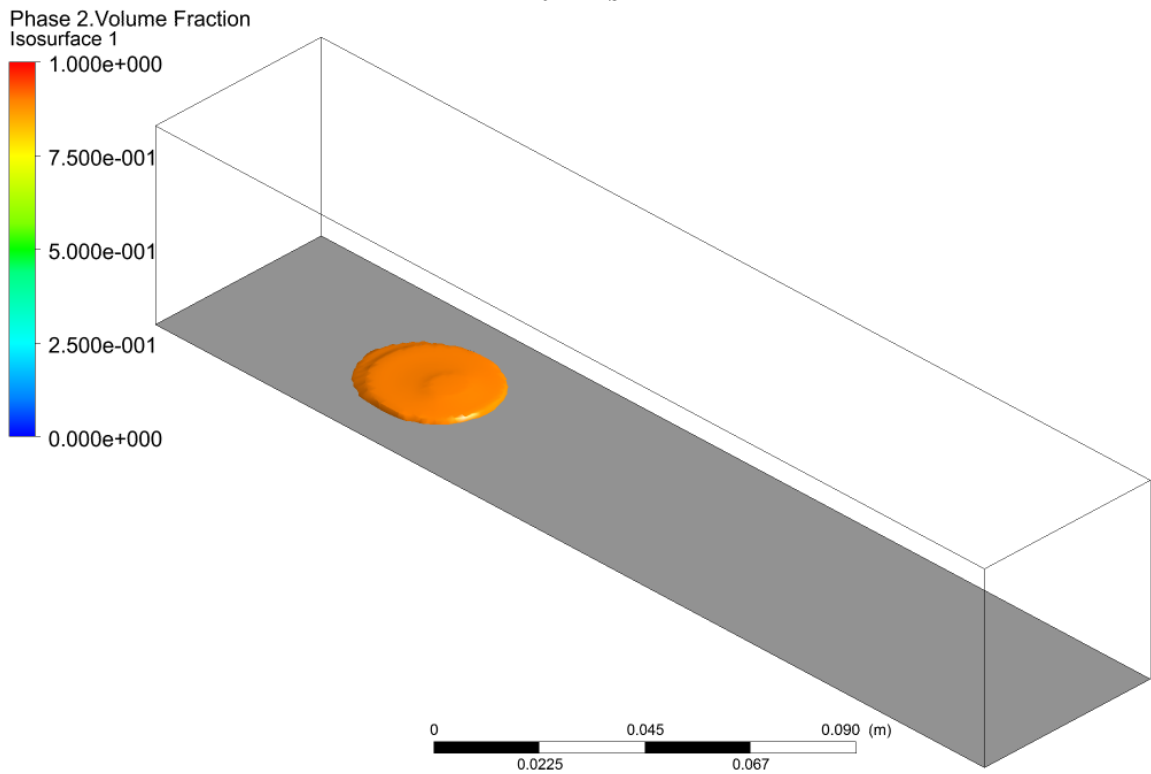
Using this domain and these boundary conditions, a simulation was performed using the viscous-laminar model. Like what was recommended, an iso-surface was created at a volume fraction of 0.9 to show the 3-D structure of the droplet. Here are the plots of this iso-surface representing the engine oil at $t = 0$ s, 0.1 s, 1 s, and 10 s. The plate was colored grey so that it would be easier to see the surface that the droplet is moving on:

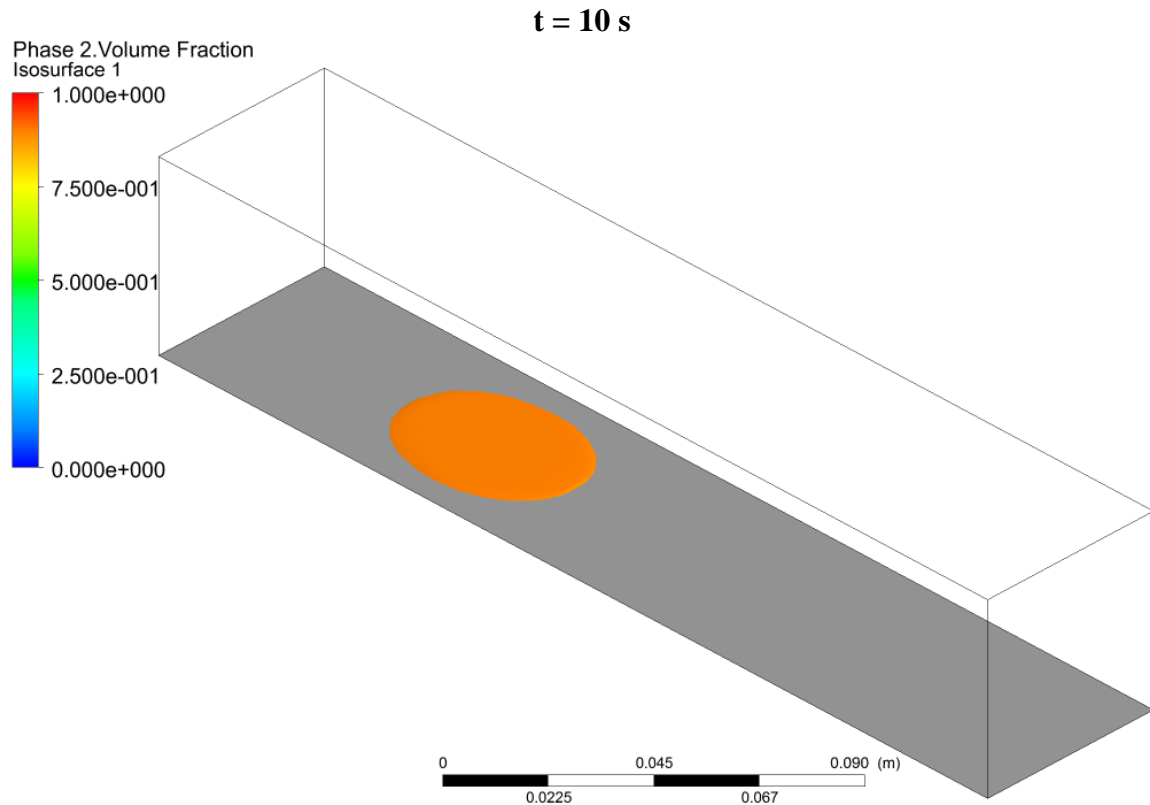


t = 0.1 s



t = 1 s





The droplet ends up spreading out quickly like what happened in the 2-D case. However, it did not stretch out nearly as far as the 2-D case since it now had the z-direction to spread out as well.

References

¹Burk, W., “Task 3,” *Homework #1*, Tempe, AZ, 2016, URL: http://www.public.asu.edu/~hhuang38/acfd_2016_hw1_ref3.pdf [cited 7 November 2016].

Appendix

Here is the MATLAB code that was used for this assignment:

```
% MAE 494/MAE 598 Project 2 Code
% This script creates the plots of the kinetic, potential, and total energy
% for both the inviscid and viscous-laminar models for a time period of 0 s
% to 25 s. Note the units for energy are J/m or N.

clear
clc

% Inviscid data and plot
Inviscid_k = dlmread('Inviscid Kinetic Energy Data.txt');
Inviscid_p = dlmread('Inviscid Potential Energy Data.txt');
Inviscid_t = dlmread('Inviscid Total Energy Data.txt');
```

```

t = Inviscid_k(:,1);
figure
plot(t,Inviscid_k(:,2),'r',t,Inviscid_p(:,2),'b',t,Inviscid_t(:,2),'g')
grid on
xlabel('Time (s)')
ylabel('Energy per Meter (J/m or N)')
title('Energy in the System for the Inviscid Case')
legend('Kinetic Energy','Potential Energy','Total Energy')

% Viscous-Laminar data and plot
Viscous_k = dlmread('Viscous Laminar Kinetic Energy Data.txt');
Viscous_p = dlmread('Viscous Laminar Potential Energy Data.txt');
Viscous_t = dlmread('Viscous Laminar Total Energy Data.txt');
figure
plot(t,Viscous_k(:,2),'r',t,Viscous_p(:,2),'b',t,Viscous_t(:,2),'g')
grid on
xlabel('Time (s)')
ylabel('Energy per Meter (J/m or N)')
title('Energy in the System for the Viscous-Laminar Case')
legend('Kinetic Energy','Potential Energy','Total Energy')

```