Project #3: Multiphase Flow

By: Ramit Gupta (No Collaboration)

MAE 494

Professor Huang

Due: November 21, 2019

Task 1:

This task utilizes the idea of VOF (Volume of Fluid) within Ansys Fluent for multiphase flow simulations. In this first task, a 2-D simulation is performed. A square bucket is modeled using the "surfaces from sketches" feature from the concept menu. There are two liquid phases used, water and air. Half of the tank is filled with liquid water simulated with a circular water droplet falling in the Y direction with gravity turned on. The total time within which the droplet falls and is mixed with the liquid water surface is set to 0.5 seconds. Screenshots of the animation at t = 0, 0.15, 0.25, 0.3, and 0.5 will be taken to see how the droplet behaves in air and once in contact with the water surface.

Geometry and Mesh:

Before modeling the tank, 2-D was selected in the properties menu. Then once the 50 cm x 50 cm square was created using surfaces from sketches, then the top outlet was set with the backflow phase set to air using named selections. The mesh element chosen size was 0.25 cm.

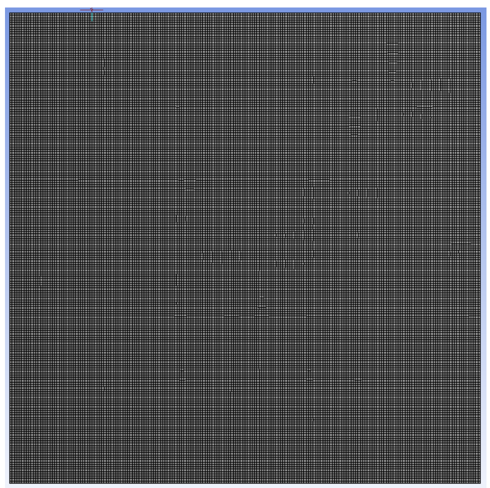

Figure 1: 2-D Geometry with Mesh

Fluent Simulation Settings:

The simulation was modeled using two separate phases first defined in the fluid selection. It is modeled as a laminar flow utilizing a transient pressure-based solver. The water droplet was centered at Y position of 40 cm and X position of 25 cm. After using standard initialization with both regions defined, the circular droplet and square water region were patched together. Then the simulation was run using a time step size of 0.0005 and 1000 total steps with 20 iterations. The results from the simulation of the volume fraction of water are below at the specified times. In the following plots, the subdomain is also shown in a green band around any liquid water.

(i) the key deliverables are contour plots of the volume fraction of water at t = 0.15 s, 0.25 s, 0.3 s, and 0.5 s. (Four separate plots.)

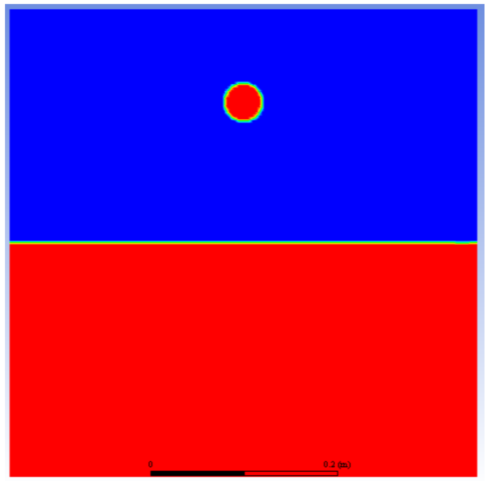

Figure 2: Contour Plot of Volume Fraction of Water at t = 0 (Initialization)

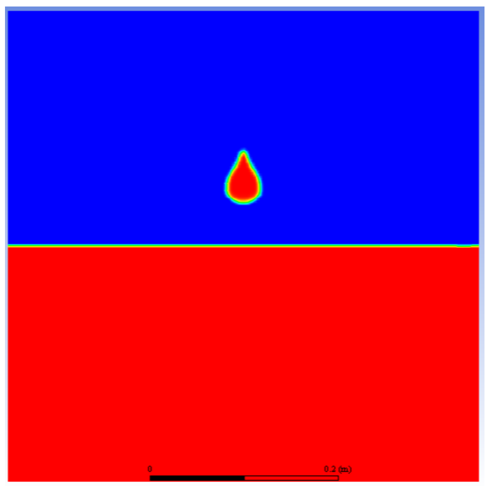

Figure 3: Contour Plot of Volume Fraction of Water at t = 0.15 s

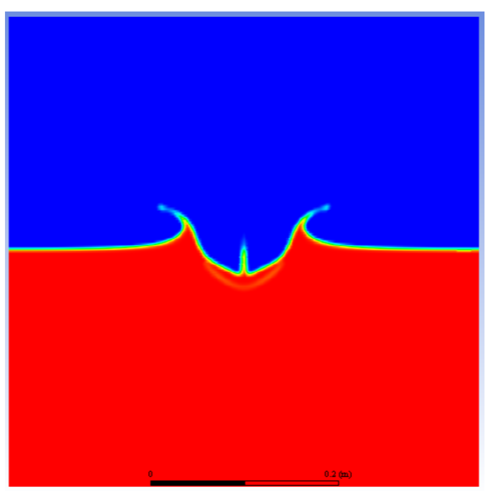

Figure 4: Contour Plot of Volume Fraction of Water at t = 0.25 s

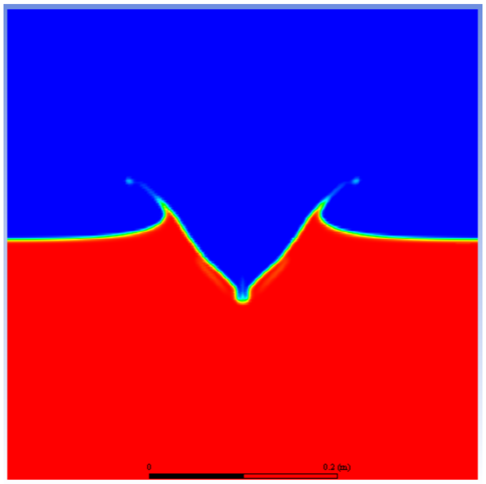

Figure 5: Contour Plot of Volume Fraction of Water at t = 0.30 s

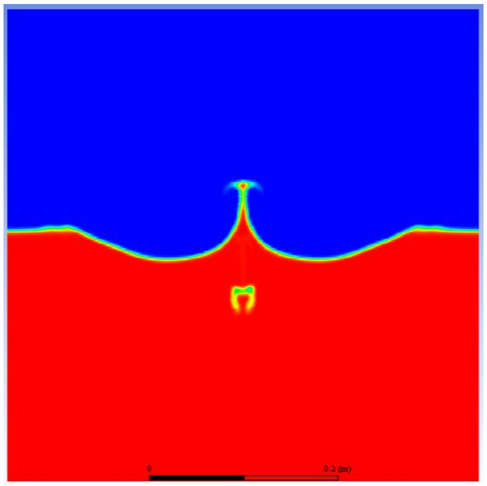

Figure 6: Contour Plot of Volume Fraction of Water at t = 0.50 s

***Task 2*:**

This task is modeled as a 3-D system because we are dealing with an incline and a droplet on the top of an inclined plate. The two fluids are air and glycerin. It is modeled as a transient simulation and a laminar system. The glycerin drop is a half sphere on the top half of the plate. The incline on the plate is 40 degrees from the horizontal, but gravity is acting directly downward at 9.81 m/s^2 in the Y direction. The effect on the gravity on the glycerin sphere and how it behaves on top of the inclined plate is seen at t = 0, 0.05, and 0.1 seconds.

Geometry and Mesh: The geometry is displayed below and is modeled in the XZ-plane with a 3-D extrusion. Element size for the original mesh was chosen to be 0.2 cm.

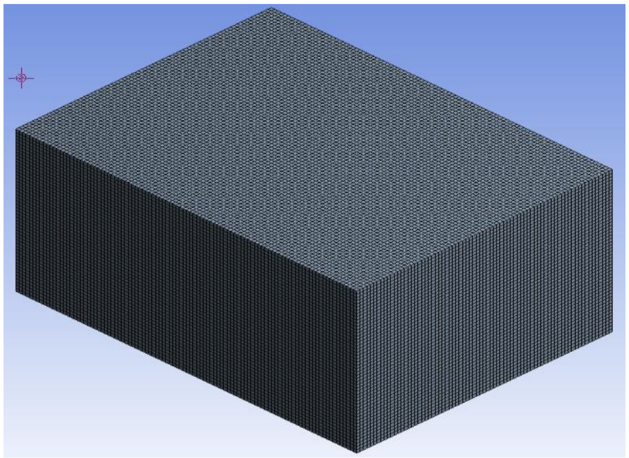

Figure 7: Geometry of System as Rectangular prism

Boundary Conditions and Computational Domain:

The system is modeled as a rectangular prism system, which serves as the boundary condition. The bottom face of the prism is the wall fluid that contains the glycerin. The bottom wall, and all other faces above it to be 10 cm in height with gravity defined in the X and Y directions. The top of the plate is also kept as an outlet face. Along with these boundary conditions, the limits of the computational domain are the length and width of the plate. Length is kept to 20 cm and the width is kept to 15 cm. This is kept in regard to the size of the sphere at radius of 2.5 cm, which is in the computational domain (inside of the boundary conditions). The sphere is in the top half of the plate and can be seen clearly with a refined mesh done in fluent seen in figure 8. With air composing the rest of the system, this chosen computational domain means the process is minimally affected.

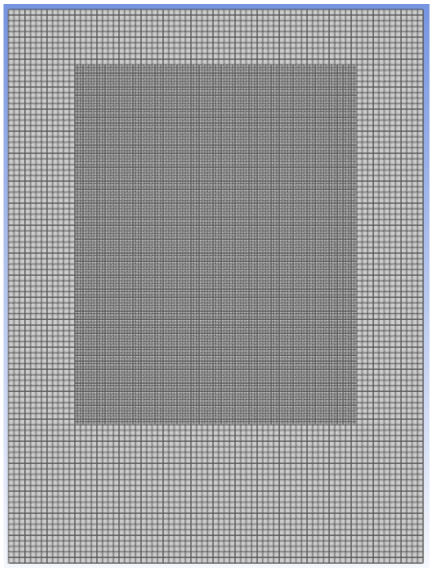

Figure 8: Geometry of Plate with refined Mesh

Fluent Simulation Settings:

The gravity is going to be components of the total gravity in the Y direction so the X gravity will be G_x = 9.81*cos(50) and Y gravity will be G_y = 9.81*sin(50). This is seen in the following figure.

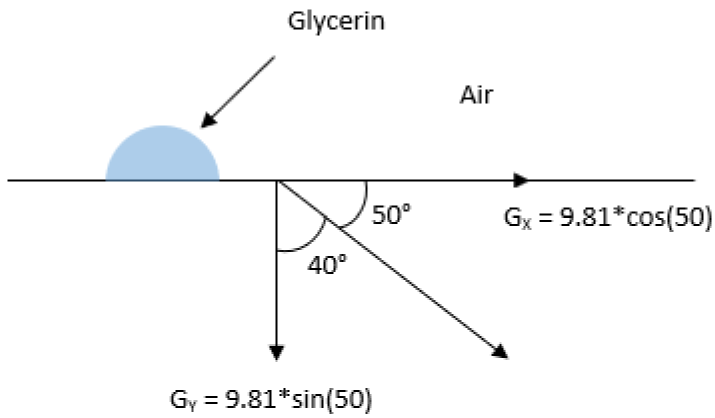

Figure 8: Figure showing Gravity components set in Fluent

The simulation was run with a 0.001 time step size with 100 total steps defining the total time of the run to be exactly 0.1 seconds with an animation playback to see the simulation at 0.05 sec as well. 20 total iterations were used.

(i) Three plots in the fashion of Fig. 2b that show the 3-D shape of the blob of glycerin at t = 0, 0.05 s, and 0.1 s.

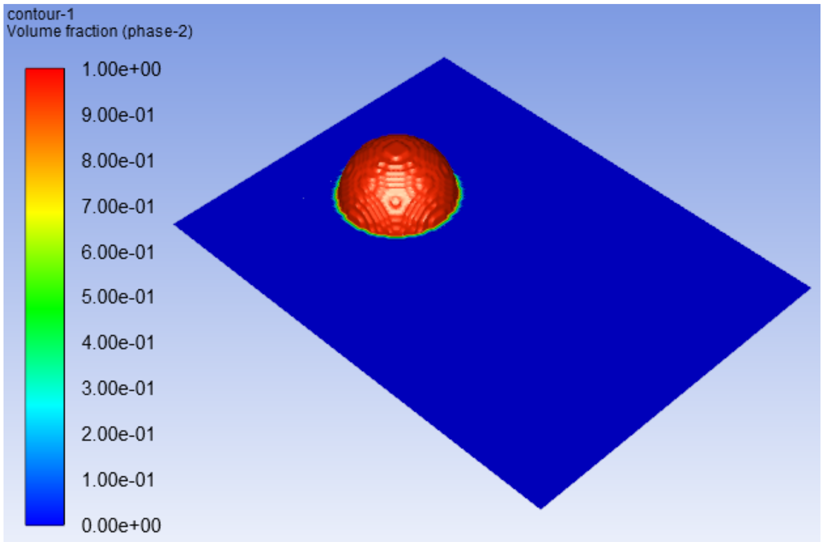

Figure 9: Initialization of the Plate with Glycerin drop on top (t = 0)

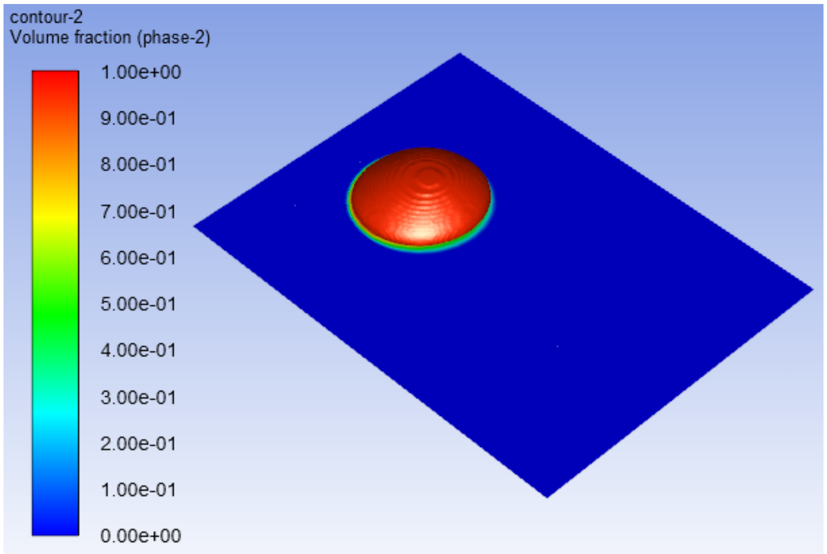

Figure 10: Plate with Glycerin drop at t = 0.05.

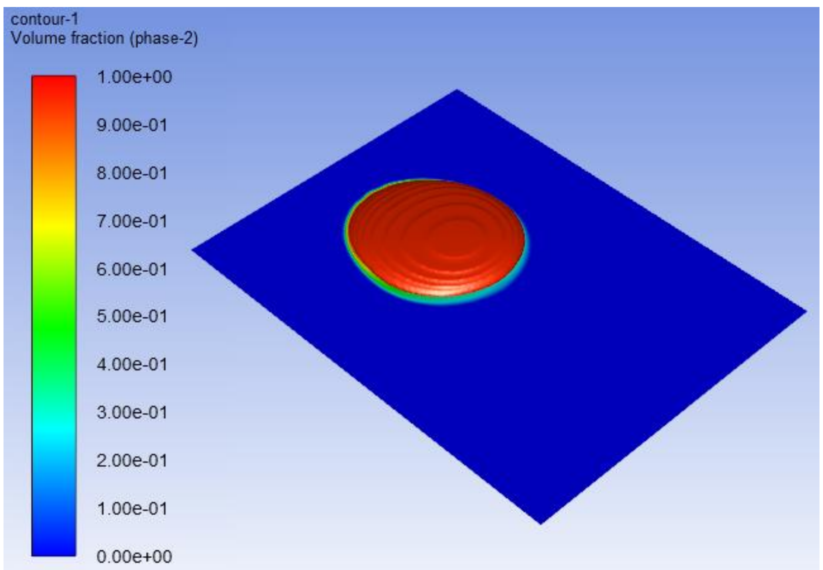

Figure 11: Plate with Glycerin drop at t = 0.1

(ii) Three contour plots of the volume fraction of glycerin on the plane of symmetry, at t = 0, 0.05 s, and 0.1 s.

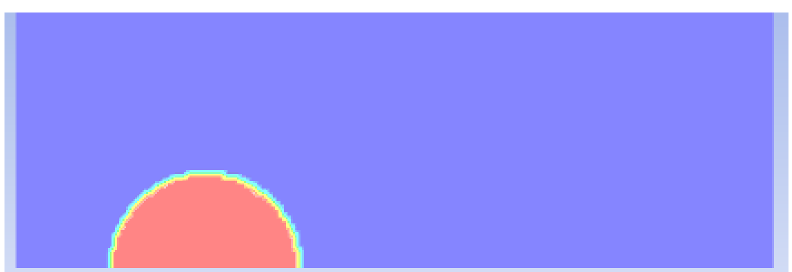

Figure 12: Volume Fraction Contour Plot at t = 0 in XY-plane

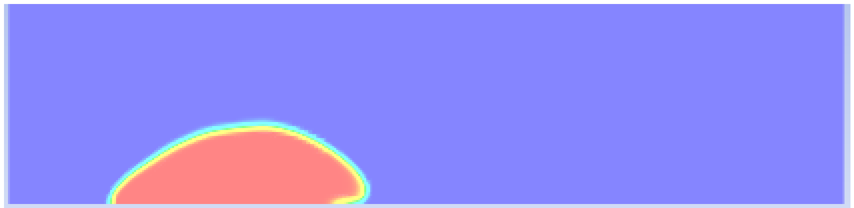

Figure 13: Volume Fraction Contour Plot at t = 0.05 in XY-plane

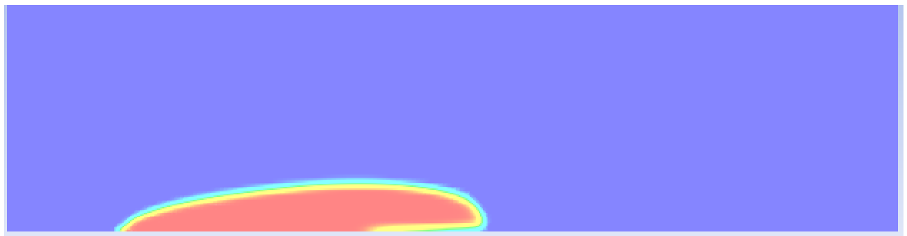

Figure 14: Volume Fraction Contour Plot at t = 0.1 in XY-plane

Task 3:

In this task, a vault with a vertical inlet opening from the bottom is simulated to show how methane gas will react with the air in a rectangular vault with three separate outlet openings restricted to gauge pressure. Air is simulated inside the vault and methane gas is simulated to represent natural gas. This is also a transient turbulent simulation with constant values kept for both fluids, and gravity set to -9.81 m/s^2 in the Y direction. Turbulence model is also used with the effect of surface tension ignored since we are dealing with two gases.

Geometry and Mesh:

The figure is a 2-D figure with three pressure outlets, (A, B, C), and a 10 m inlet opening. The left corner of the tank is set to the origin and the tank is 50 m high by 120 m wide. The mesh size used was 0.5 m for both part a) and part b).

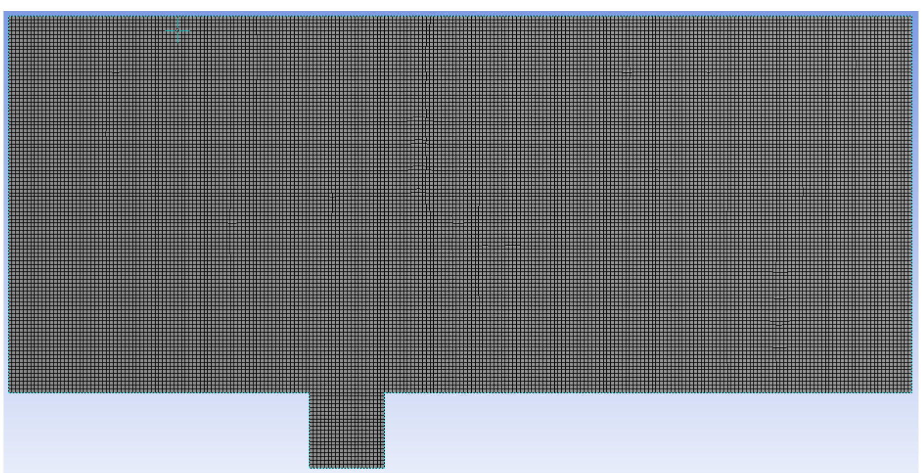

Figure 15: 2-D Geometry with Mesh

Fluent Simulation Settings:

Three outlets were chosen and the inlet gauge pressure is kept to 50 Pa with methane as the gas being released from the top. Gravity is set in the -Y direction and the simulation is run with a time step of 0.01 with 200 total steps. 20 iterations were used once the entire domain was checked to be filled with air. Part a) has only a pressure outlet for side A, but Part b) also has a linear velocity profile modeled to notice the difference in direction of natural gas flow compared to Part a).

(a) Contour plots of the volume fraction of methane at t = 7 s and t = 10 s.

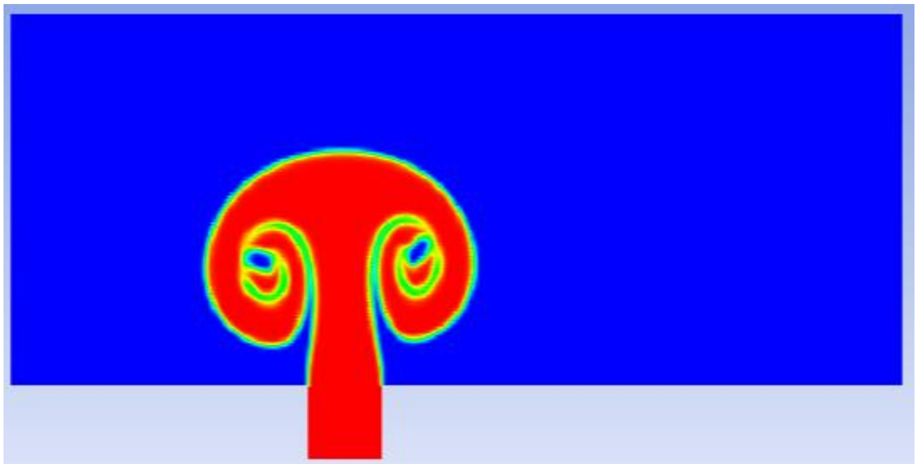

Figure 16: Contour plot of VF of methane at t = 7

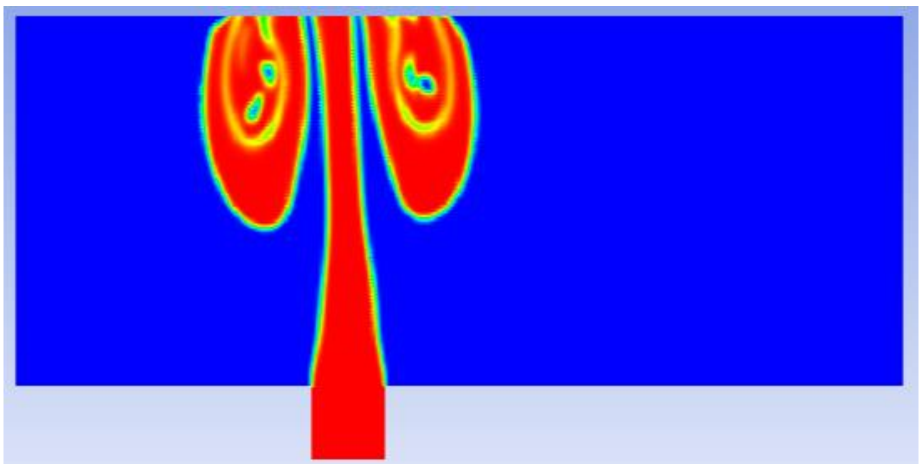

Figure 17: Contour plot of VF of methane at t = 10

(b) Contour plots of the volume fraction of methane at t = 9 s and t = 12 s with velocity profile.

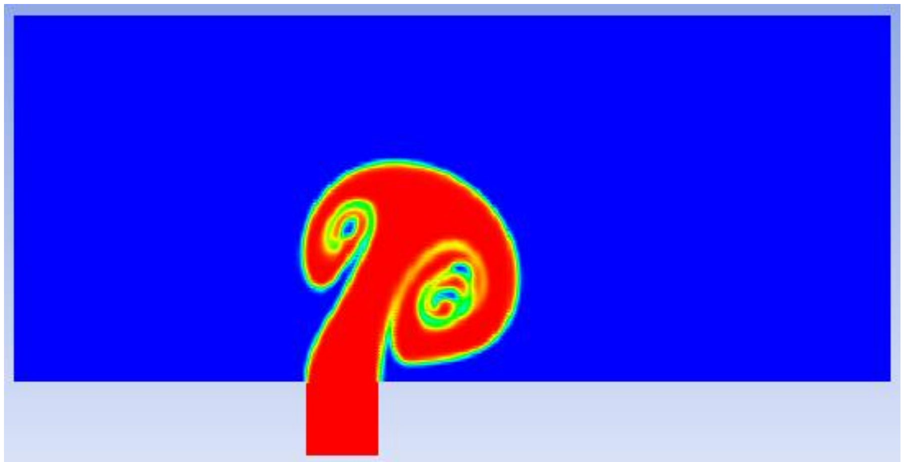

Figure 18: Contour plot of VF of methane at t = 9

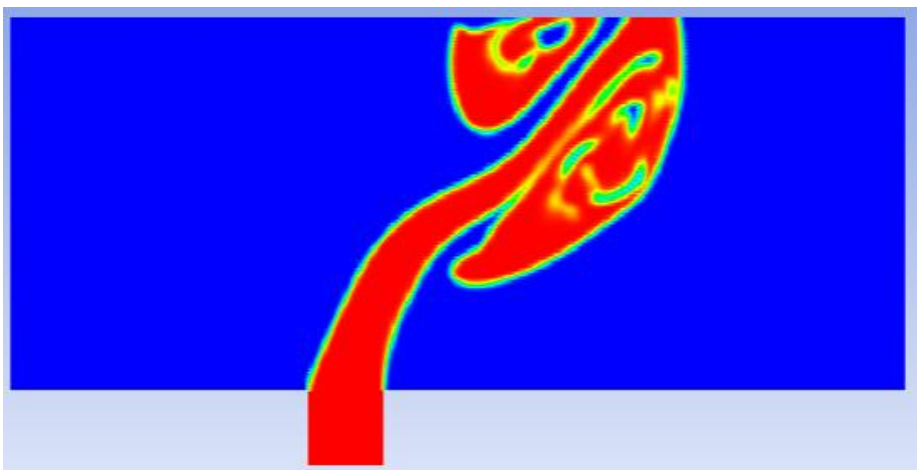

Figure 19: Contour plot of VF of methane at t = 12