

MAE 560 – Applied CFD – Fall 2022

Project 2 – – Multiphase flow; Flow with interface

Aishwarya Ledalla

10/28/2022

Statement of Collaboration

No collaboration

All Tasks

Instructions:

- All tasks in this project should use the VOF model in ANSYS-Fluent for multiphase flow simulations. In all tasks, set the density of the individual phase of fluid (water, air, etc.) to constant and run the simulation with pressure-based solver. Since none of the tasks involves thermodynamic processes, Energy equation can be turned off. The choice of laminar or turbulence model will be given in the individual tasks. For Task 1, set operating density method to mixture-averaged. For all other tasks, set it to minimum-phase-averaged (which is the default). [Note: In older (pre-2020) versions of Ansys, the default is “not set”, which is equivalent to mixture-averaged in the current version of Ansys.]

Task 1

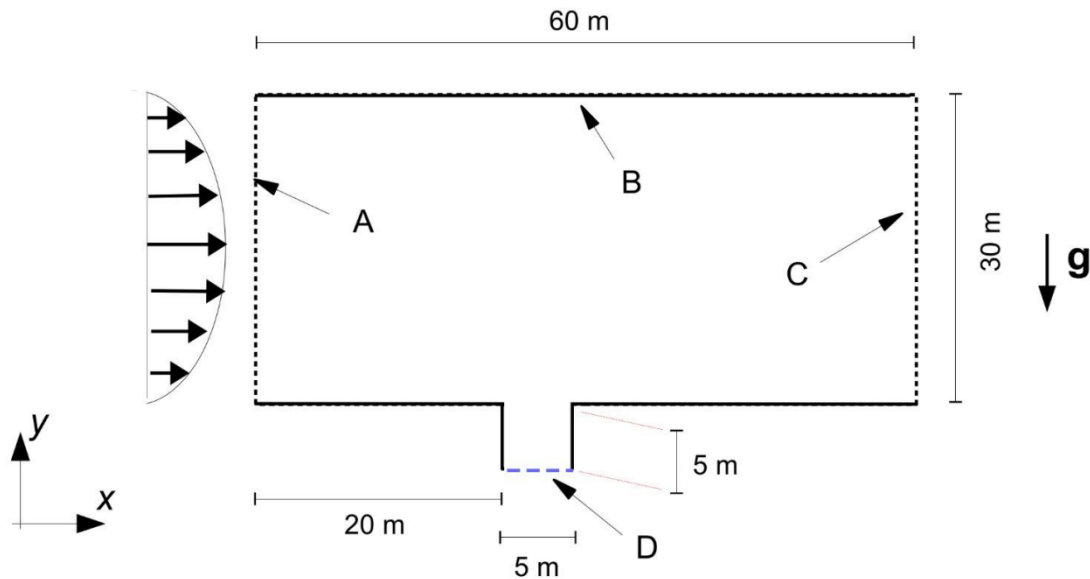


Figure 0: Computational Domain for Task 1 [1].

Objective:

Simulate leaking of natural gas from an underground vault into open air, in a pure 2D setting.

Instructions:

- In this task, we simulate the leaking of natural gas from an underground vault into open air, in a pure 2-D setting. The computational domain is as shown in Fig. 1. A pipe at bottom is connected to a pressurized reservoir of methane (representing natural gas). This task includes two transient simulations. In both, use the default constant values of density and viscosity from Fluent database for air and methane, and set gravity to the regular $g = -9.81 \text{ m/s}^2$ in the vertical direction (the “y direction” in Fig. 1). Use turbulence k-omega model with default setting. Since both phases are gases, the effect of surface tension can be ignored.

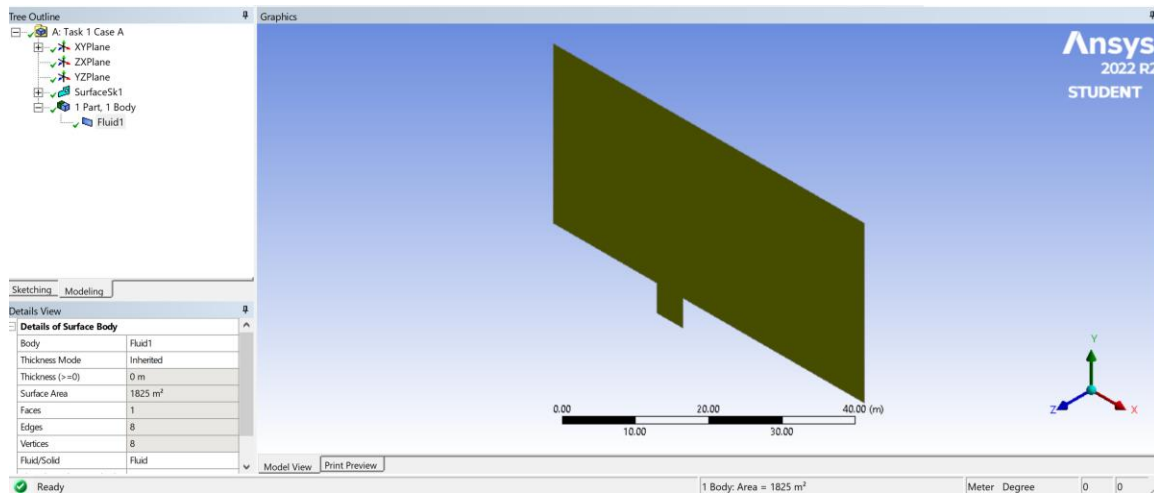


Figure 1: Geometry of Computational Domain (Task 1).

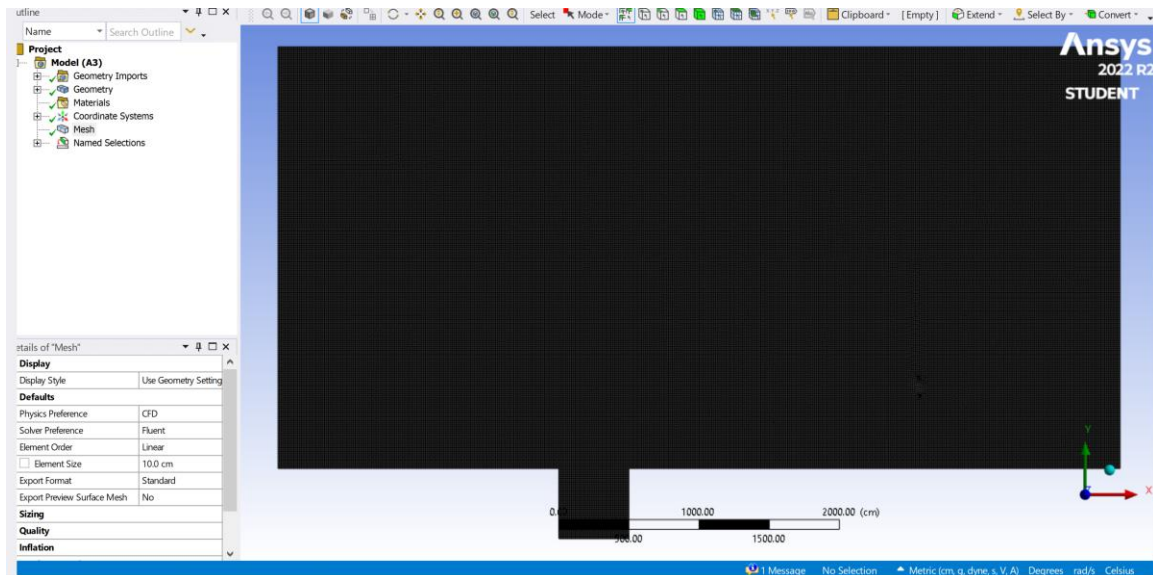


Figure 2: Mesh of Computation Domain (Task 1).

- Case A:** Set the two boundaries marked by A and C to pressure outlet with zero gauge pressure. Both are open to air. The top boundary marked by B is set to wall but with zero shear stress. (This emulates a “frictionless wall.”) The boundary marked by D is set as a pressure inlet with gauge pressure = 50 Pa. Methane is pumped through inlet D into the domain. (All other unnamed boundaries are regular wall with no-slip boundary condition, i.e., the default setting for wall.) At $t = 0$, fill the entire domain (including the bottom pipe) with air. Initialize the system with gauge pressure = 0 and velocity = 0. For the turbulence

parameters, initialize turbulence kinetic energy (k) = $1 \text{ m}^2 \text{ s}^{-2}$, and specific dissipation rate (ω) = 1 s^{-1} . Perform the transient simulation to $t = 6 \text{ s}$.

- (D1)

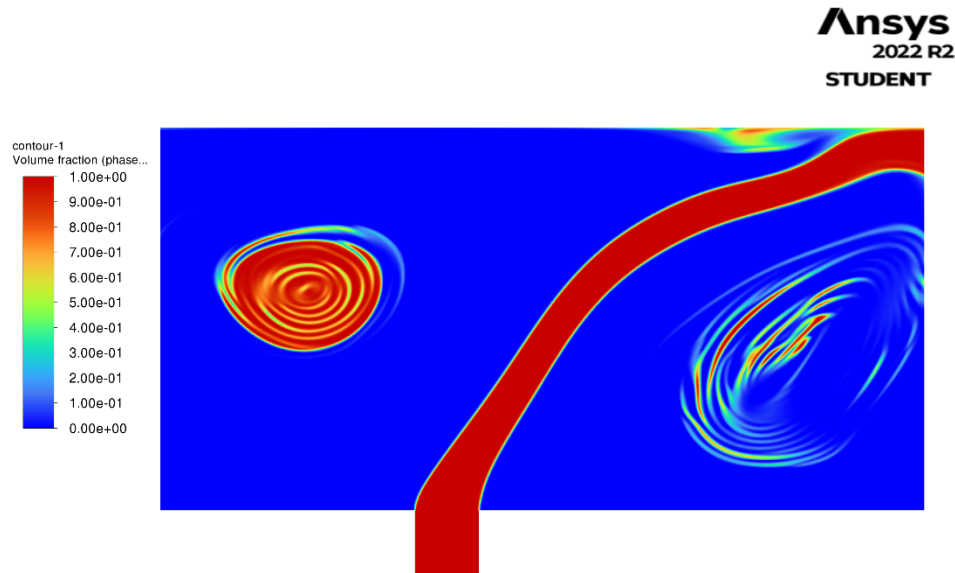


Figure 3: Contour Plot of Volume Fraction of Methane at $t = 3 \text{ s}$ (D1).

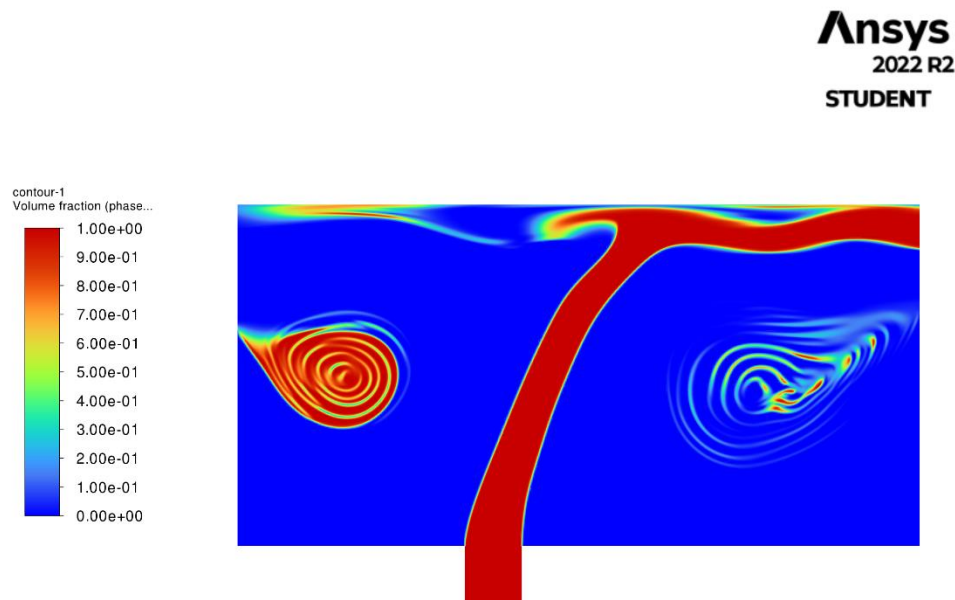


Figure 4: Contour Plot of Volume Fraction of Methane at $t = 6 \text{ s}$ (D1).

- Case B:** Use the same setting as Case A except that the gauge pressure at inlet D is increased to 150 Pa. In addition, the left boundary marked by A is replaced by a velocity inlet, with an imposed velocity profile for the x-velocity given by $u = 0.6y - 0.02y^2$ where u is in m/s and y in m. (Otherwise, gauge pressure is set to zero at this inlet.) This gives a parabolic profile with $u = 0$ at the ground ($y = 0$) and top of the domain ($y = 30$ m), and u attains the maximum of 4.5 m/s at $y = 15$ m. (See a sketch in Fig. 1.) Air is pumped into the domain through inlet A. At $t = 0$, initialize the system in the same way as Case I. Perform the transient simulation to $t = 6$ s.

(D2)

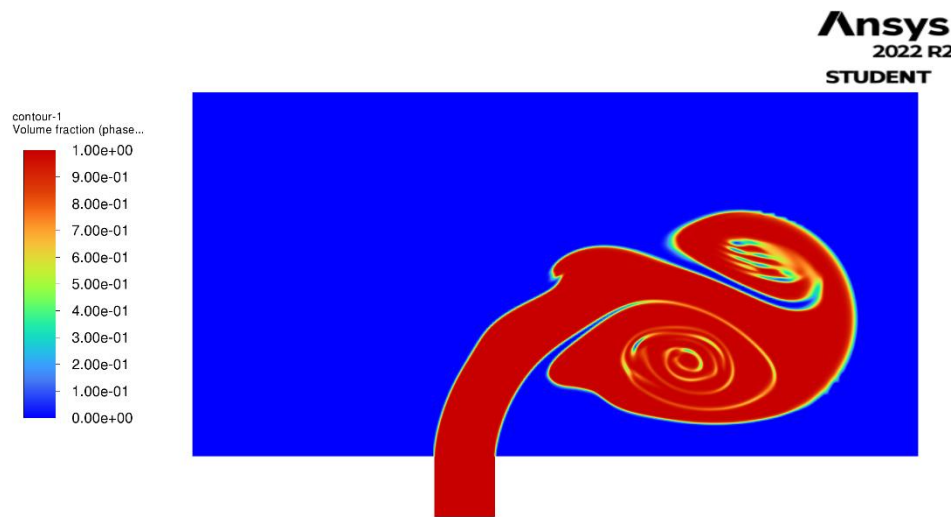


Figure 5: Contour Plot of Volume Fraction of Methane at $t = 6$ s (D2).

(D3) Mesh resolution, time step size, maximum number of iterations per time step for Cases A and B:

For both cases, I chose a mesh with an element size of 0.1 m and a time step of 0.01 s. The mesh resolution was high enough to capture the behavior of the swirling pattern of the leaking methane. A higher resolution would have taken a long time and lower resolution would not have captured enough details of the fluid interaction. I initially chose a time step of 0.5 but it was too small as it was not even showing the methane entering the computational domain. For case A, I chose 20 iterations per time step while I chose 10 for case B. From case A, I realized 20, which is the default, was too many iterations and settled on half of that number to lower the

runtime of the simulation while maintaining the accuracy of the calculations. After seeing the contour plot, I think the number of iterations should have been adjusted to 15 instead of 10 for a higher accuracy of the simulations results.

Task 2

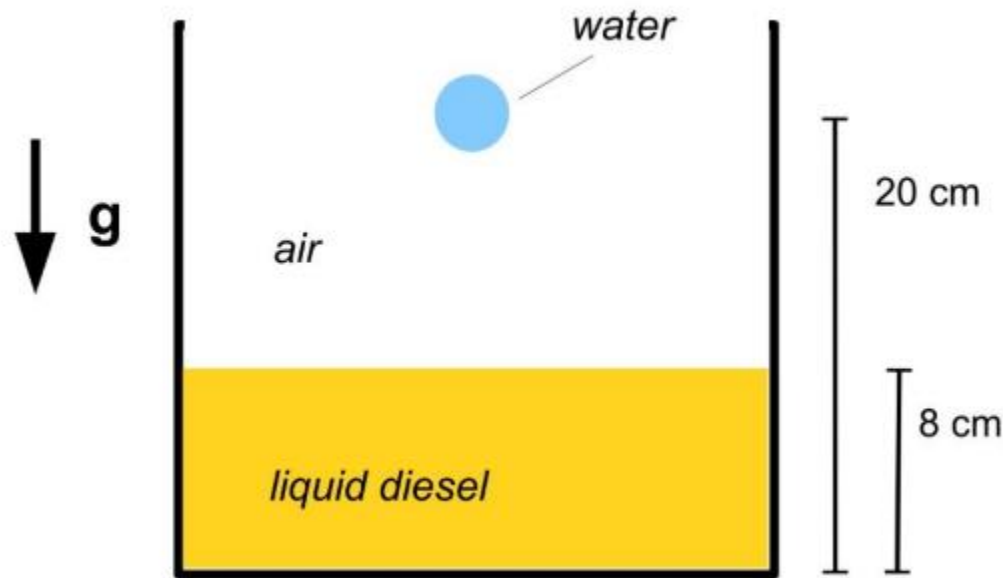


Figure 6: Computational Domain for Task 2 [1].

Objective:

Simulate process of a falling water droplet colliding with flat surface of liquid diesel, in a pure 2D setting.

Instructions:

- In this task, we simulate the process of a falling water droplet colliding with a flat surface of liquid diesel, in a pure 2-D setting. The geometry of the system is a simple 25 cm x 25 cm square bucket that is open (to air) at the top and with the other three sides being walls, as shown in Fig. 2. At $t = 0$, the bucket is partially filled with liquid diesel (“diesel-liquid” in Fluent database) to the depth of 8 cm, and the rest filled with air. (See Fig. 2.) In addition, a circular droplet with radius of 1.5 cm (diameter of 3 cm) is placed in the middle of the bucket, with its center placed 20 cm above the floor. More precisely, if the coordinate of the lower-left corner of the bucket is $(x, y) = (0,0)$, the center of the droplet is $(x, y) = (12.5 \text{ cm}, 20 \text{ cm})$. For the transient simulation, set gravity to the regular $g = -9.81 \text{ m/s}^2$ in the y-direction as indicated in Fig. 2. Initialize the system with zero gauge pressure and zero velocity. Use Laminar model and turn on surface tension modeling. For

the interface between water and air, use the default value of surface tension coefficient (under liquid water) from Fluent database. Otherwise, ignore surface tension effect for the interface between water and liquid diesel, and between liquid diesel and air. At $t > 0$, the water droplet will begin to fall and eventually impact the surface of liquid diesel. Perform the transient simulation to $t = 0.2$ s.

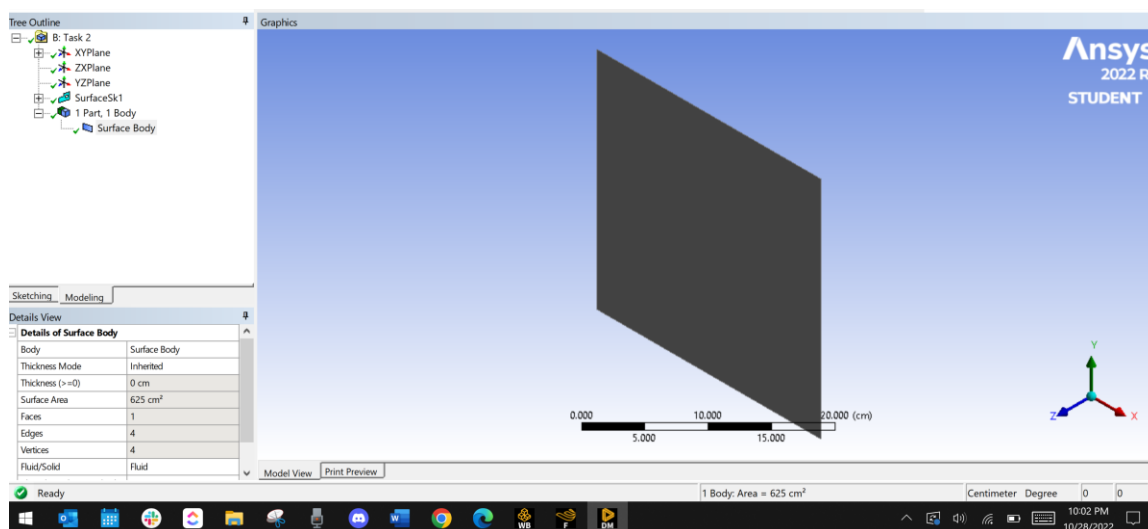


Figure 7: Geometry of Computational Domain (Task 2).

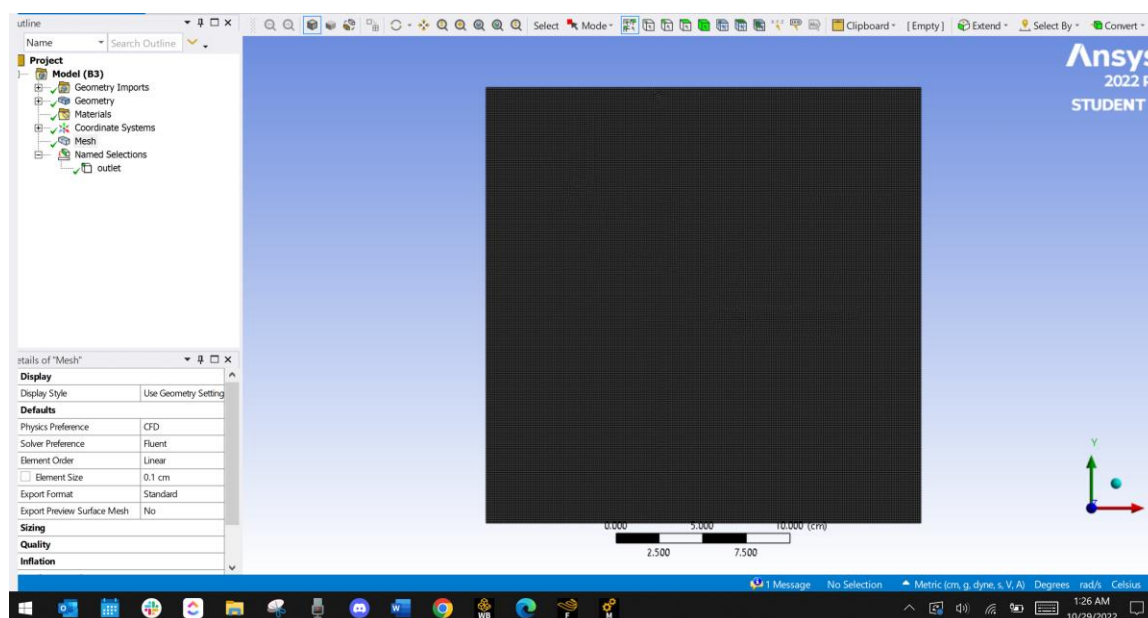


Figure 8: Mesh of Computation Domain (Task 2).

(D4) Mesh resolution, time step size, maximum number of iterations per time step:

I chose a mesh with an element size of 0.1 cm since the dimensions of the bucket are 25 x 25 cm. I thought it would give it enough resolution to show detail. I initially chose a time step size of 0.05 but it was too low as the contour plot was not showing any change in the system. I then changed it to 0.0008 s with the default 20 iterations per time step. Since I had to simulate three phases, I used the following *custom field function*:

$$CF = 1 * VF_{air} + 3 * VF_{water} + 5 * VF_{diesel} .$$

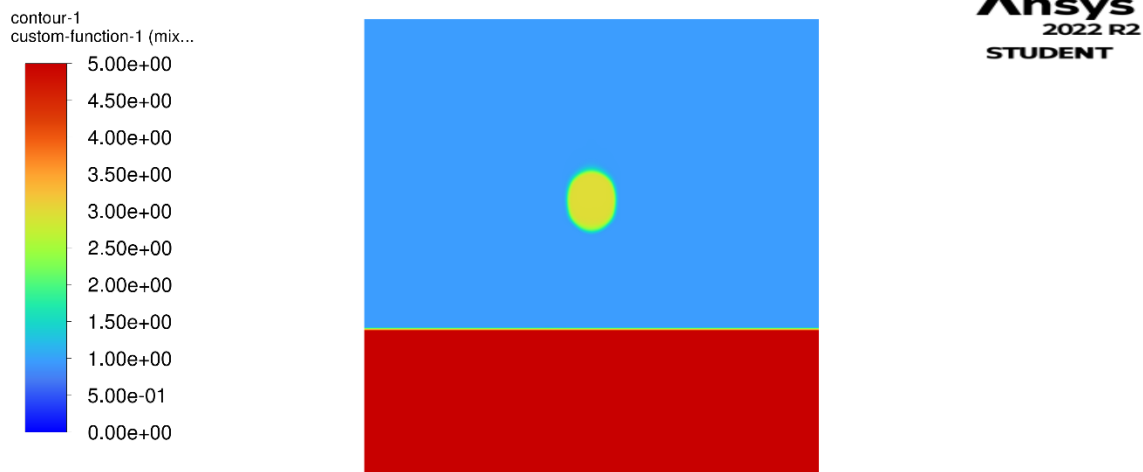
(D5)

Figure 9: Contour Plot of Custom Field Function CF at t = 0.1s (D5).

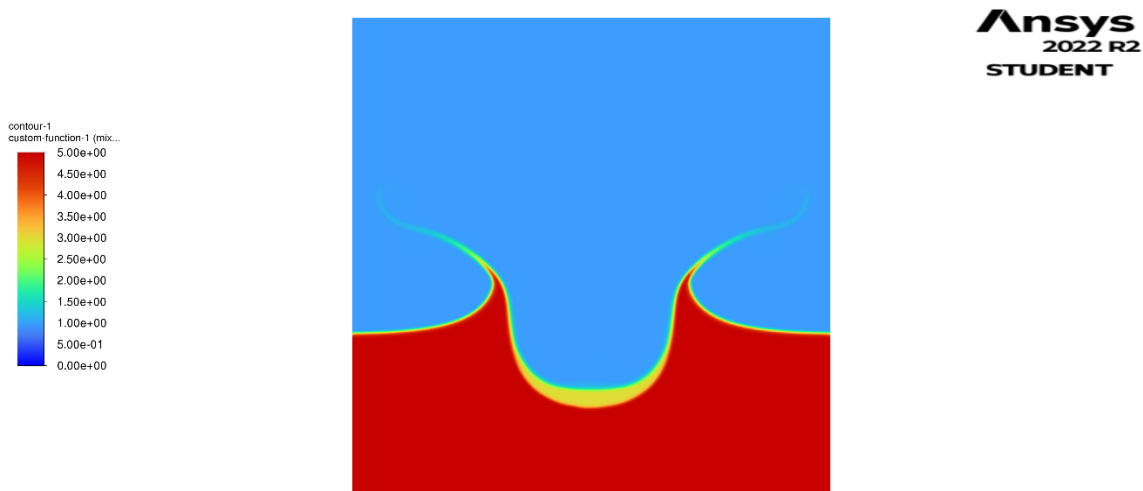
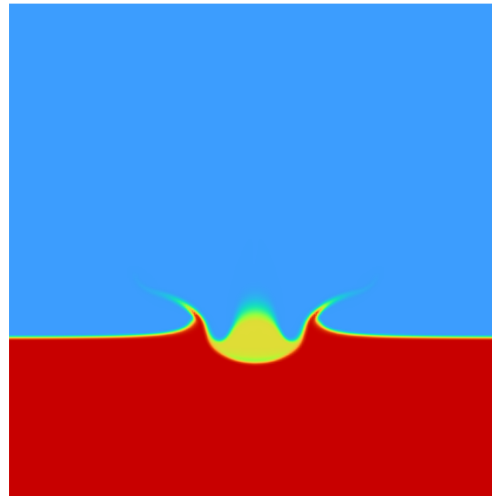
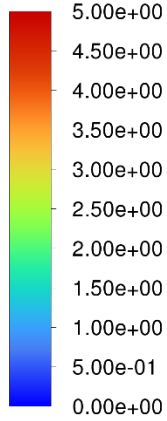


Figure 10: Contour Plot of Custom Field Function CF at t = 0.16s (D5).

contour-1
custom-function-1 (mix...



Ansys
2022 R2
STUDENT

Figure 11: Contour Plot of Custom Field Function CF at $t = 0.2\text{s}$ (D5).

Task 3

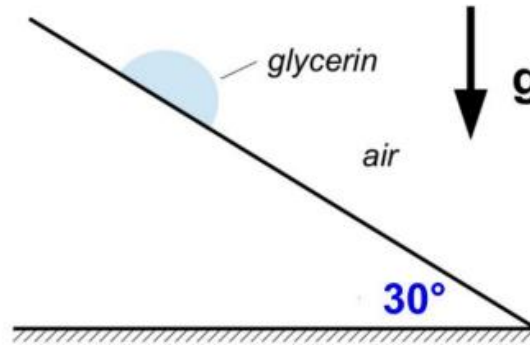


Figure 12: Computational Domain for Task 3 [1].

Objective:

Simulate the temporal evolution of a glycerin droplet in air on an inclined plate.

Instructions:

- Consider a 3-D system with an inclined plate that forms a 30° angle with the ground. At $t = 0$, a droplet of glycerin is placed on the plate, and it is shaped like a hemisphere with a radius of 2 cm (diameter of 4 cm). The droplet is surrounded by open air. Gravity is the regular $g = -9.81 \text{ m/s}^2$ in the vertical direction. Figure 3a shows the cross-sectional view of the system along the vertical plane that cuts through the center of the droplet. Figure 3b is the 3-D isometric view (explained below) of the initial state of the system, with the bottom plate shown in blue. This task will simulate the temporal evolution of the droplet. In the actual physical system, the droplet of glycerin is surrounded by open air without top or side “wall” boundaries. In the numerical simulation, one needs to set a computational domain and specify appropriate boundary conditions to emulate this physical condition. Appropriate mesh resolution and time step size should be used to ensure that the result is robust. Use Laminar model and use the default constant values for the density and viscosity of glycerin and air. Turn on surface tension modeling and set surface tension coefficient for the interaction between glycerin and air to a constant of 0.06 N/m. Perform a transient simulation to $t = 0.2 \text{ s}$.

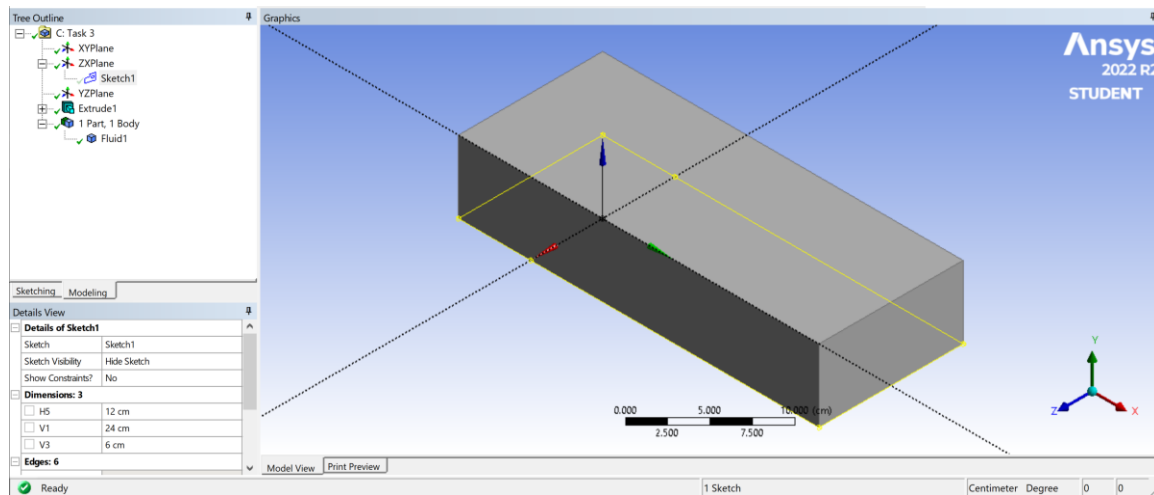


Figure 13: Geometry of Computational Domain (Task 3).

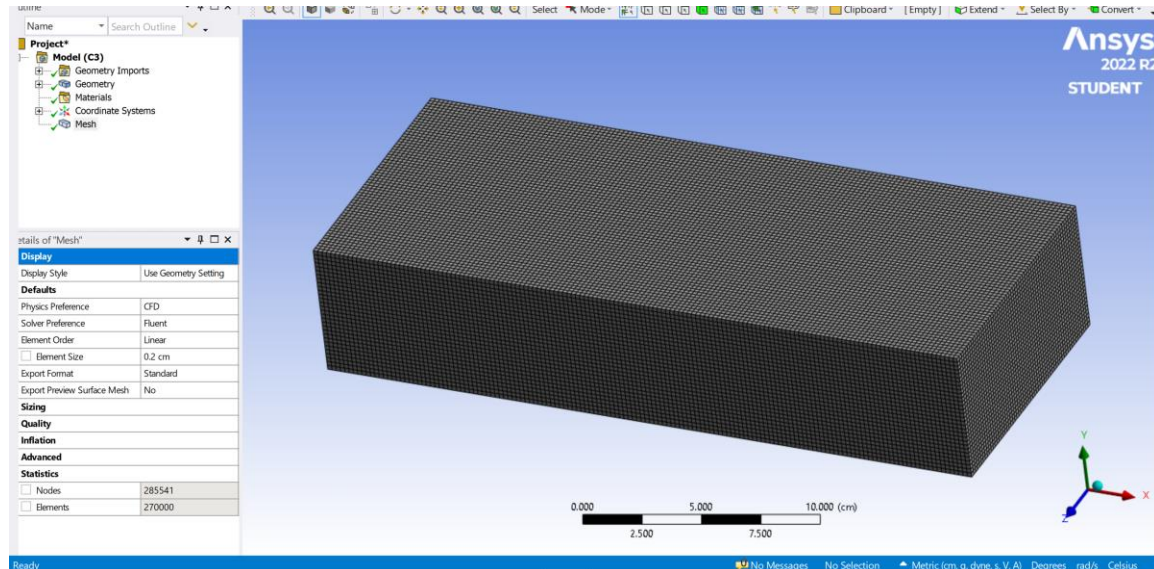


Figure 14: Mesh of Computation Domain (Task 3).

(D6) Computation Domain, mesh resolution, time step size, maximum number of iterations per time step:

I chose a rectangular computational domain to encompass both the glycerin blob and the inclined plate. The dimensions I chose were ____ so that the blob would fit on the plate as it slides down the inclined plate due to gravity. I rotated the XY axis by 30° instead of rotated the plate to simplify the simulation. I adjusted the gravity components accordingly.

Gravity in x, y, and z directions when the XY plane is rotated by 30° :

$$g_x = g \sin(\theta^\circ) = (9.81) \sin(30^\circ) = 4.905 \text{ m/s}^2$$

$$g_y = g \sin(\theta^\circ) = (-9.81) \cos(30^\circ) = -8.49571 \text{ m/s}^2$$

I chose 0.2 cm for the element size of the mesh as it was the smallest size yielding the number of nodes under the node limitation of the student version of ANSYS. My time step was 0.0008 again since the computational domain were similar in size to the one in task 3 as well as the loads involved. I used the default 20 iterations/time step as standard. I chose $VF_{\text{glycerin}} = 1$ for iso-surface of the glycerin drop.

(D7)

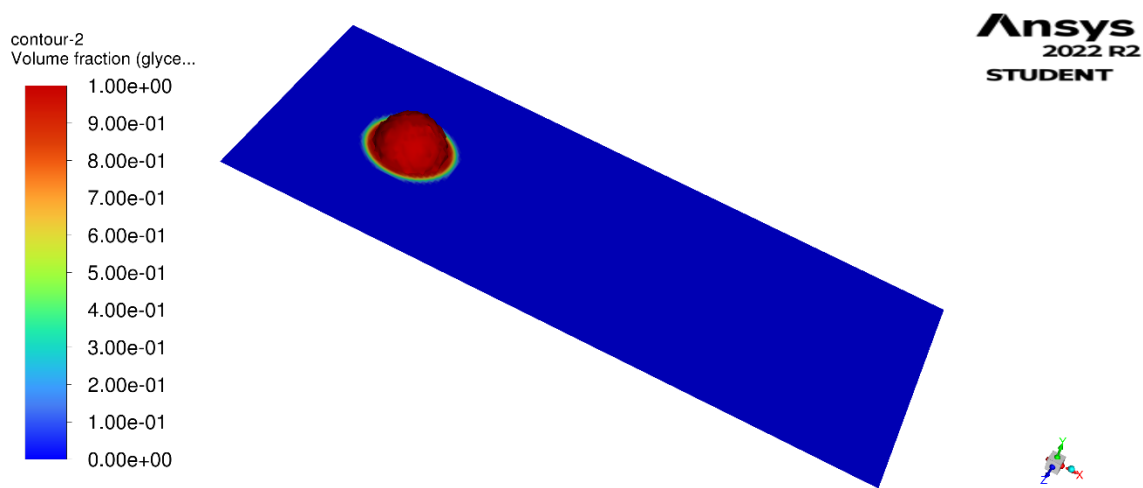


Figure 15: Glycerin blob at 0 (D7).

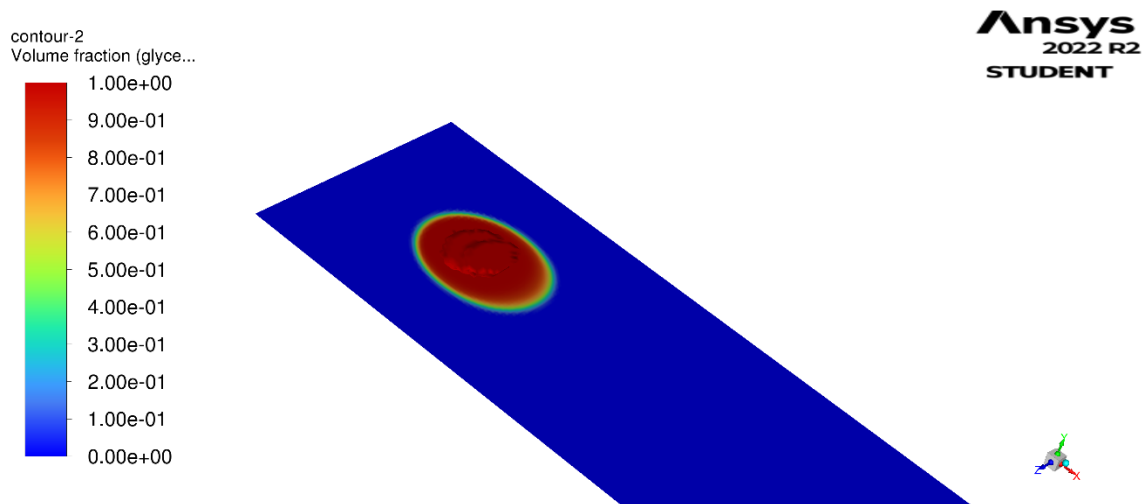


Figure 16: Glycerin blob at 0.08s (D7).

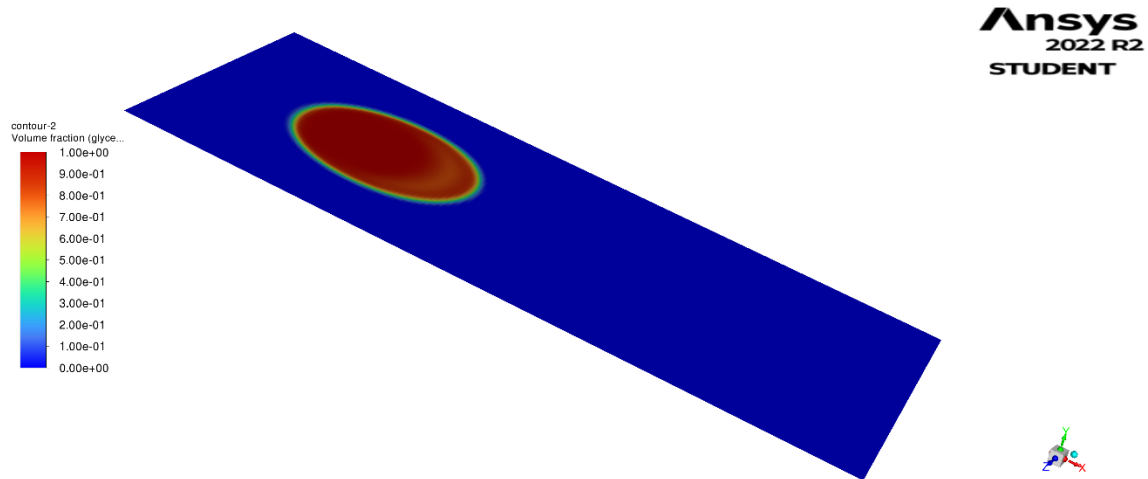


Figure 17: Glycerin blob at 0.2s (D7).

(D8)



Figure 18: Contour plot of Volume Fraction of Glycerin on Plane of Symmetry at 0s (D8).

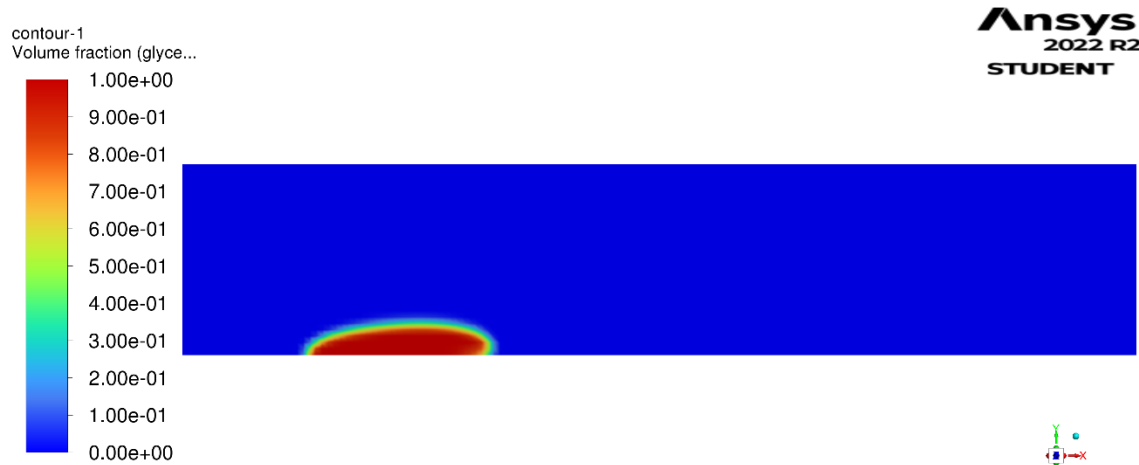


Figure 19: Contour plot of Volume Fraction of Glycerin on Plane of Symmetry at 0.08s (D8).

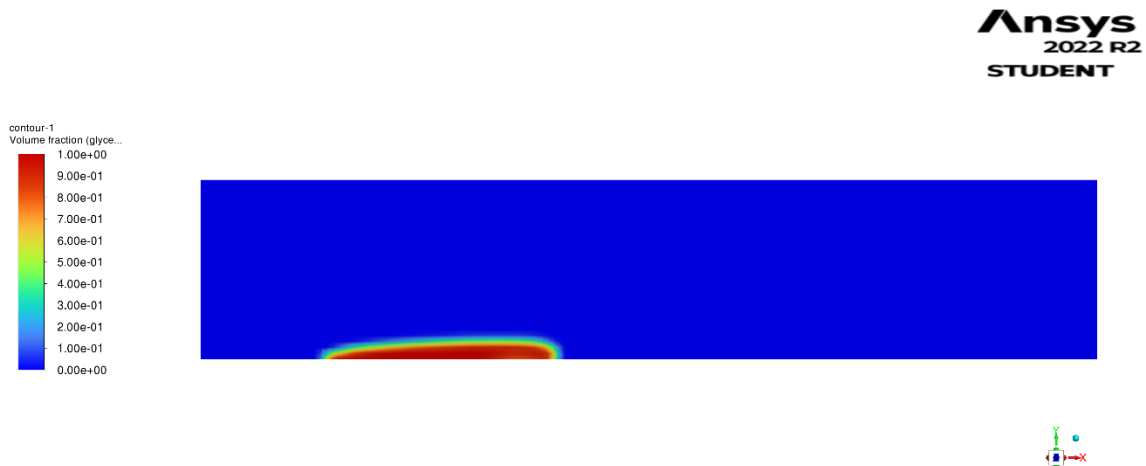


Figure 20: Contour plot of Volume Fraction of Glycerin on Plane of Symmetry at 0.2s (D8).

Task 4

Objective:

Simulate fluid drainage from a kitchen sink prototype with water and engine oil.

Instructions:

- For simplicity, we turn off surface tension for this task. Use k-omega model with default setting. Consider a prototype of a kitchen sink that consists of a hemispherical basin with a radius of 25 cm, and a cylindrical drainage pipe at bottom that is 5 cm long with a radius of 2 cm. The 3D view of the system is shown in Fig. 4a, and a cross-sectional view along the plane of symmetry in Fig. 4b. The top of the basin and bottom of the drainage pipe are both open to air. Gravity is the regular $g = -9.81 \text{ m/s}^2$ in the vertical direction as indicated in Fig. 4a.

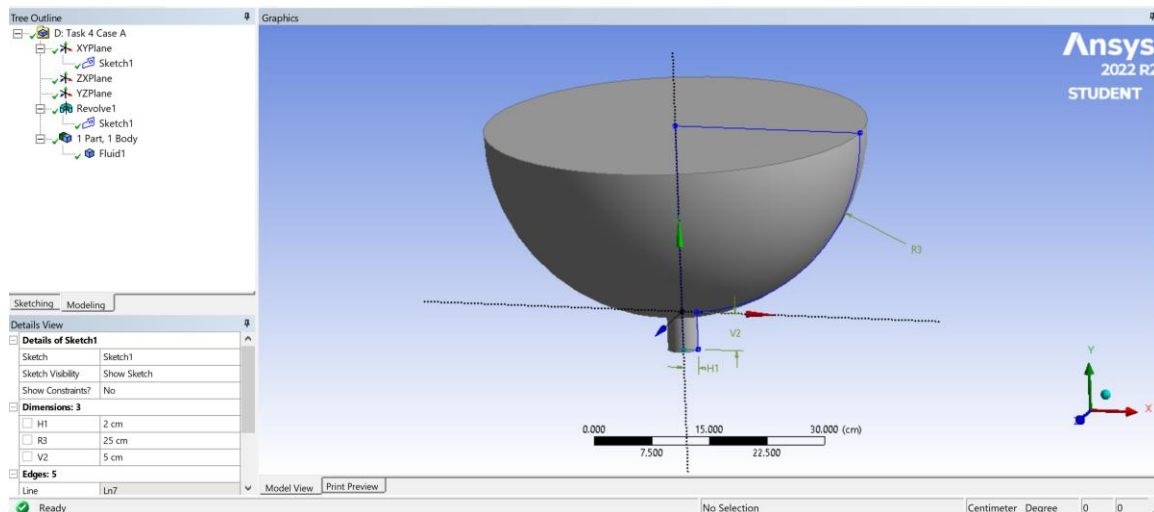


Figure 13: Geometry of Computational Domain (Task 4)

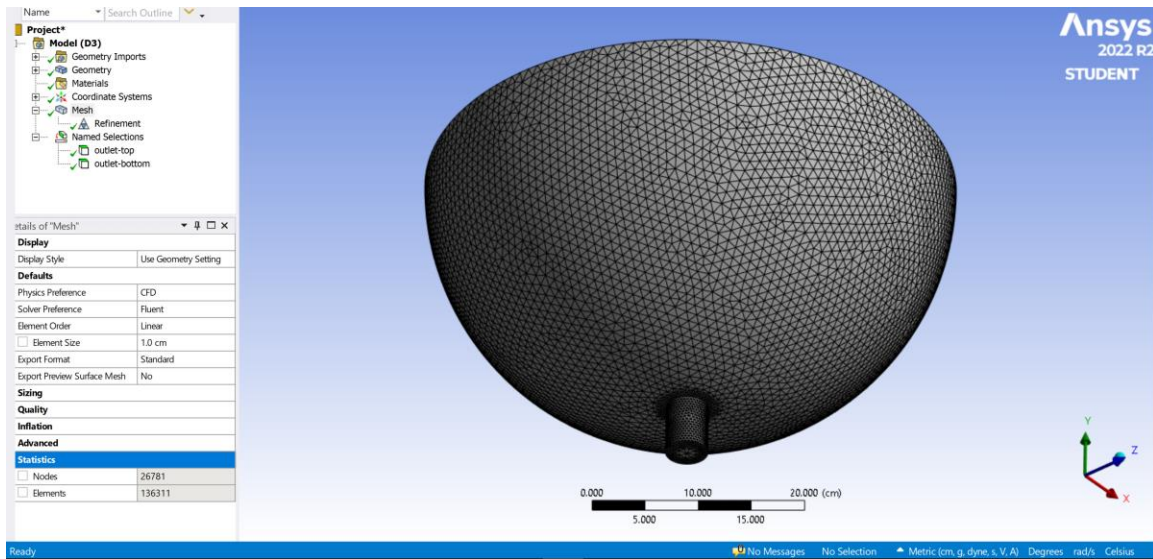


Figure 21: Mesh with Mesh Refinement of Computation Domain (Task 4).

- **Case A:** Consider an initial state with the kitchen sink (including the bottom pipe) filled with water to the level of 5 cm from the top, as shown in Fig. 4b (water and air are red and blue, respectively). From this initial state, perform a transient simulation to the time, $t = tA$, at which half of the water in the initial state is drained out of the system.
- **Case B:** Repeat the setting in Case A but replace water with engine oil (using the physical properties from Fluent database.) Run the simulation to the time, $t = tB$, at which half of the engine oil is drained out of the system.

(D9) Boundary conditions, mesh resolution, time step size, maximum number of iterations per time step:

For the boundary conditions at the top were pressure inlet since liquids usually go into a sink. At this inlet, air with $VF = 1$ was at zero-gauge pressure. I made the bottom a pressure outlet with air ($VF = 1$) at zero-gauge pressure. I chose a mesh resolution of 1 cm with a mesh refinement on the pipe at the bottom. The time step was 0.05 with 10 maximum number of iterations.

(D10)

$$M_{water} = 19.4 \text{ kg} \quad \Rightarrow \quad \frac{M_{water}}{2} = 9.7 \text{ kg}$$

$$\boxed{t_A = 10s}$$

(D11)

$$M_{water} = 22.4 \text{ kg} \quad \Rightarrow \quad \frac{M_{water}}{2} = 11 \text{ kg}$$

$$\boxed{t_A = 13s}$$

Resources:

*** All instructions sections directly from [1]. ***

[1] MAE 560/460 Applied CFD, Fall 2022, Project 2 – Multiphase flow, Flow with interface instructions