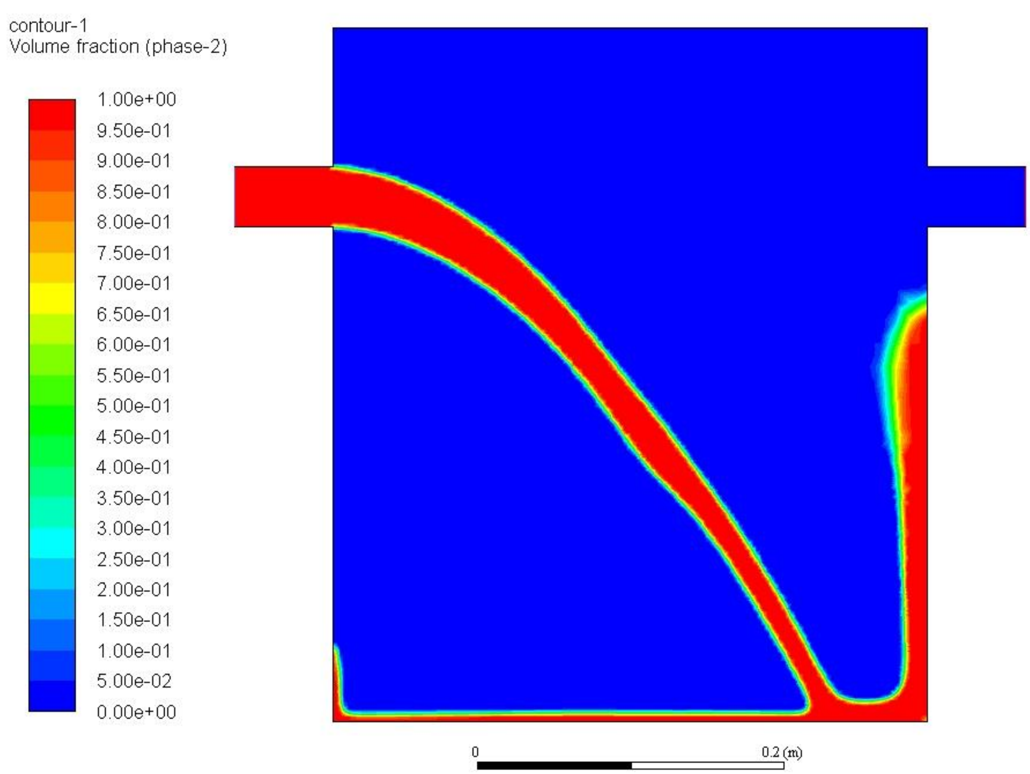
APPLIED COMPUTATIONAL FLUID DYNAMICS-PROJECT-3

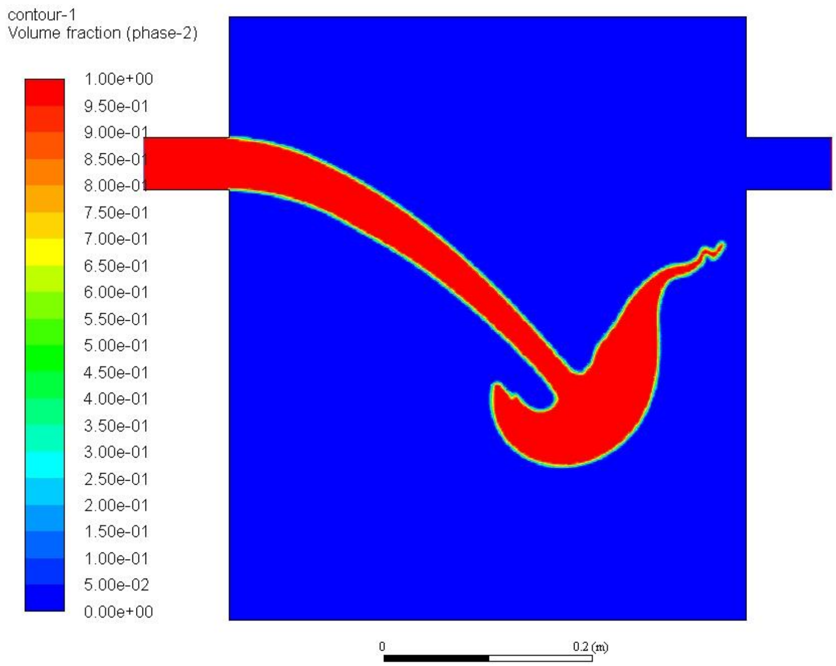
BY SAI CHAITANYA MANGAVELLI

- Common Setup Data:
  1) Mesh – Proximity and Curvature with Refinement of 2.
  2) Double Precision and second order for methods in Solver.

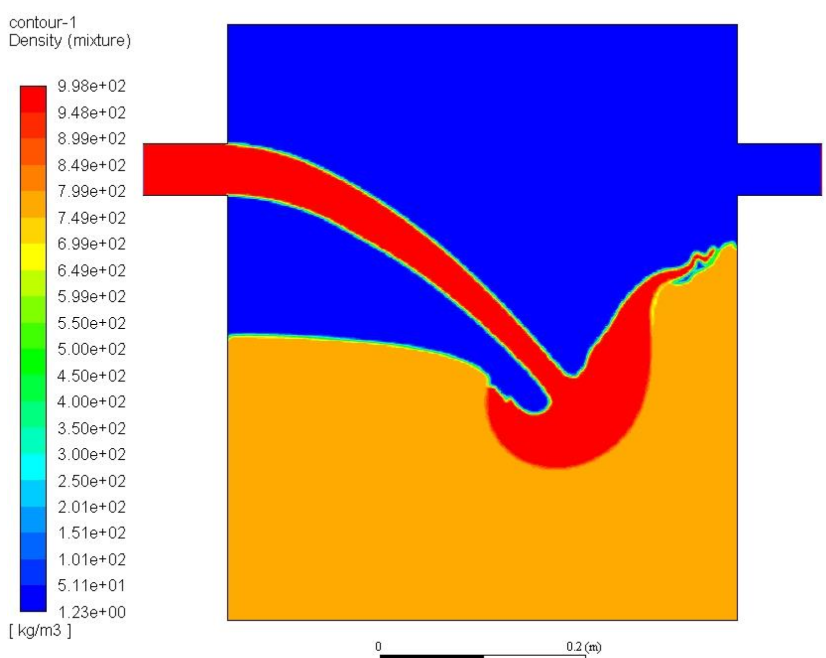
- Task-1-a:
  1) Mesh Data: 61,000 (Nodes)
  2) Solver Settings: K-Epsilon (Standard), Multi-Phase Model with air and water, Courant Number: 0.5-1.2, Time Step Size: 0.001s
  3) Deliverables:
     - Contour Plot of volume fraction of water at 0.4s



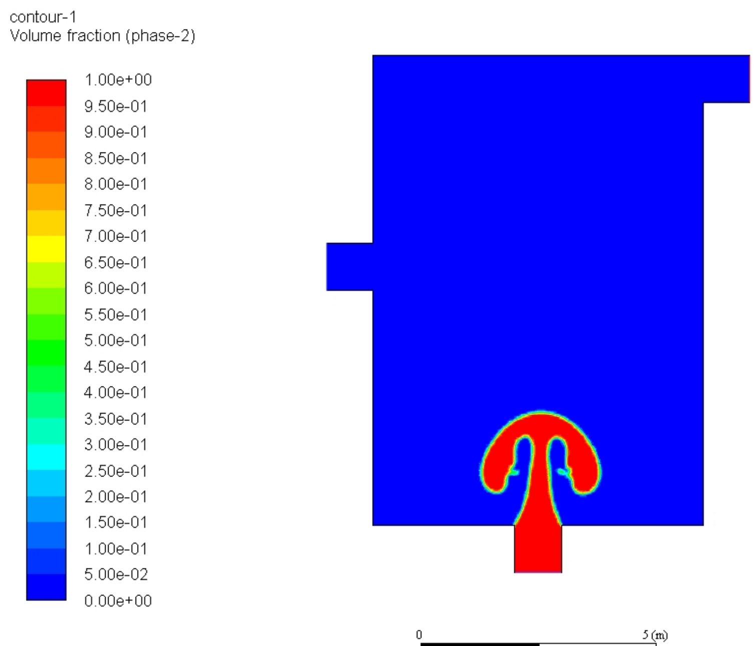
- Task-1-b:
    1) Solver Setup: K-Epsilon(Standard), Multiphase Model with air, water, liquid-kerosene, Courant Number: 0.5-1.3, Time Step Size: 0.001s
    2) Deliverables:
        (a) Contour Plot of Volume Fraction of water at t=0.4s



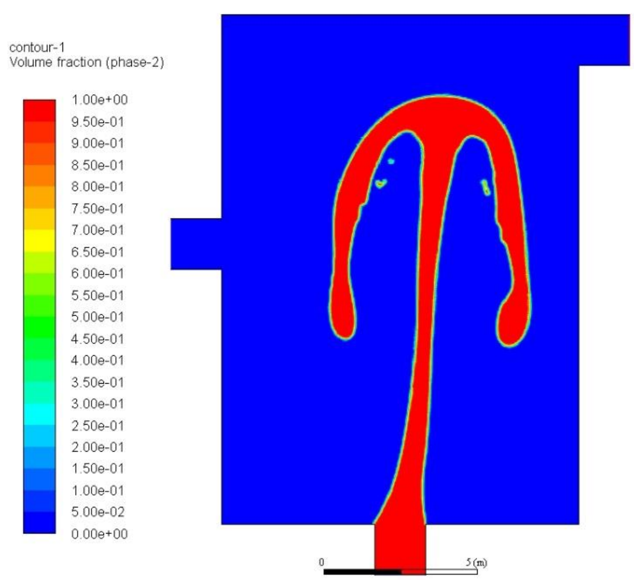
        (b) Contour Plot of total density of mixture at t=0.4s



- Task-2-a
    1) Mesh Data: 52,184(nodes)
    2) Solver Settings: K-Epsilon(Standard), Multiphase Model-air, methane, Courant Number- 0.08-0.95, Time Step Size – 0.001s
    3) V-air at inlet = 1m/s
    4) V-methane at inlet-2m/s
    5) Deliverables:
        (a) Volume Fraction of methane at 2s



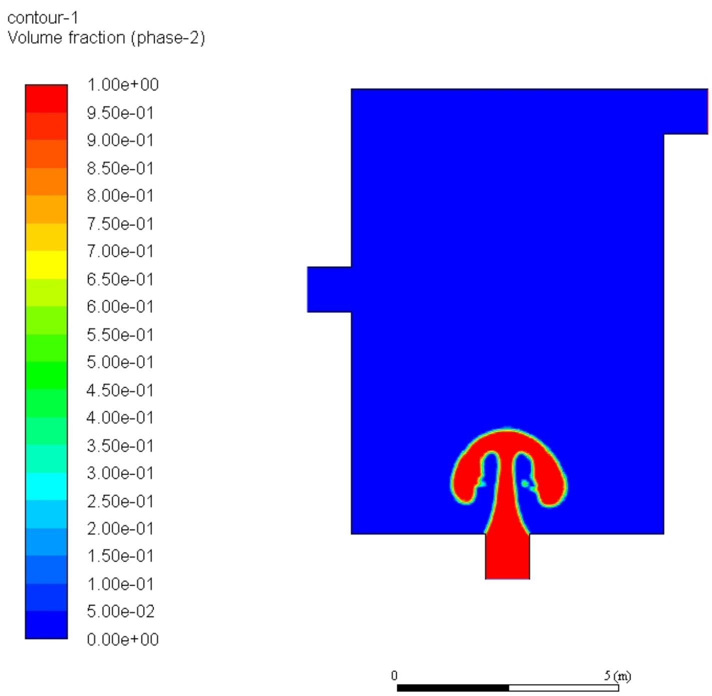
        (b) Volume Fraction of methane at t=5s



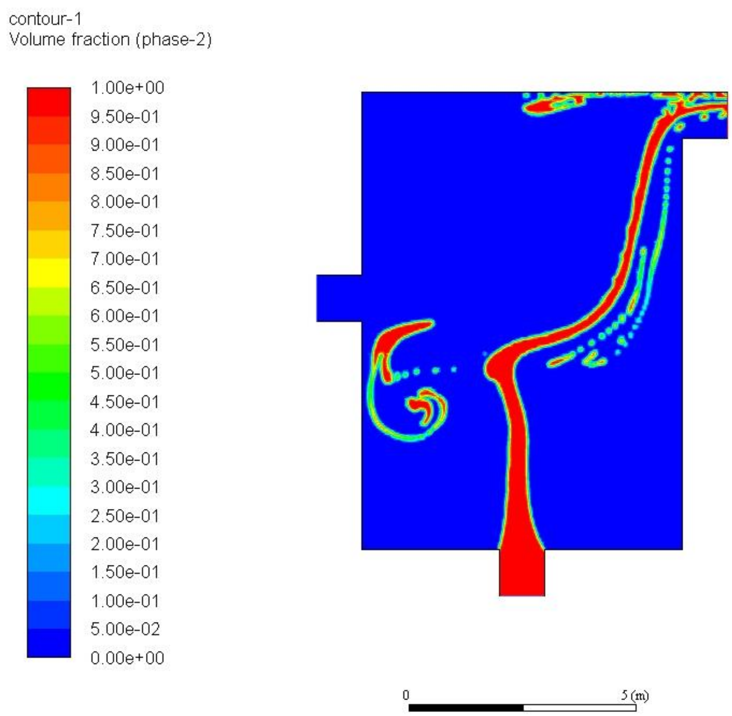
- Task-2-b:
  1) V-inlet of air: 5m/s

  2) Rest all settings are maintained the same, Courant Number-0.06-1.4, Time Step Size – 0.001s

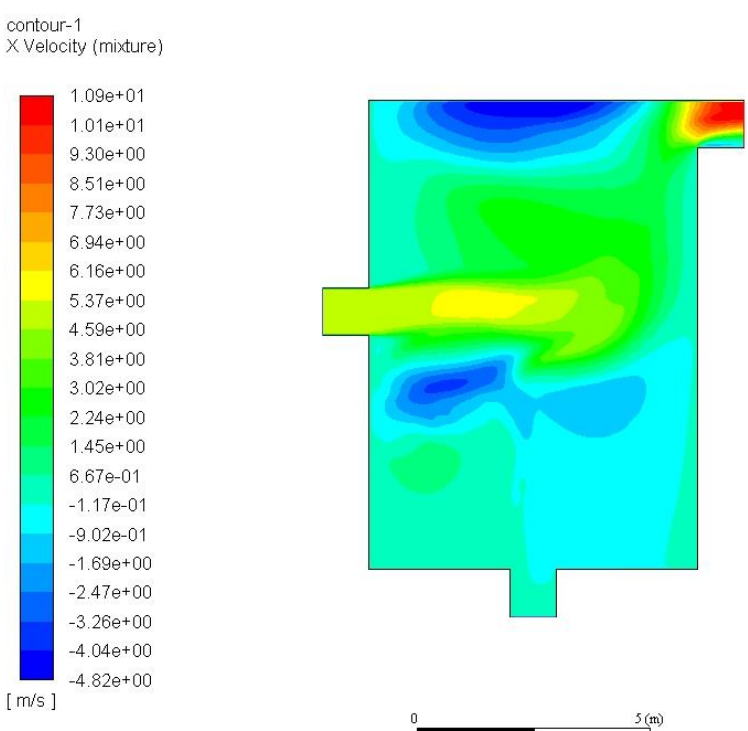
  2) Deliverables:
     (a) Volume Fraction of methane at 2s



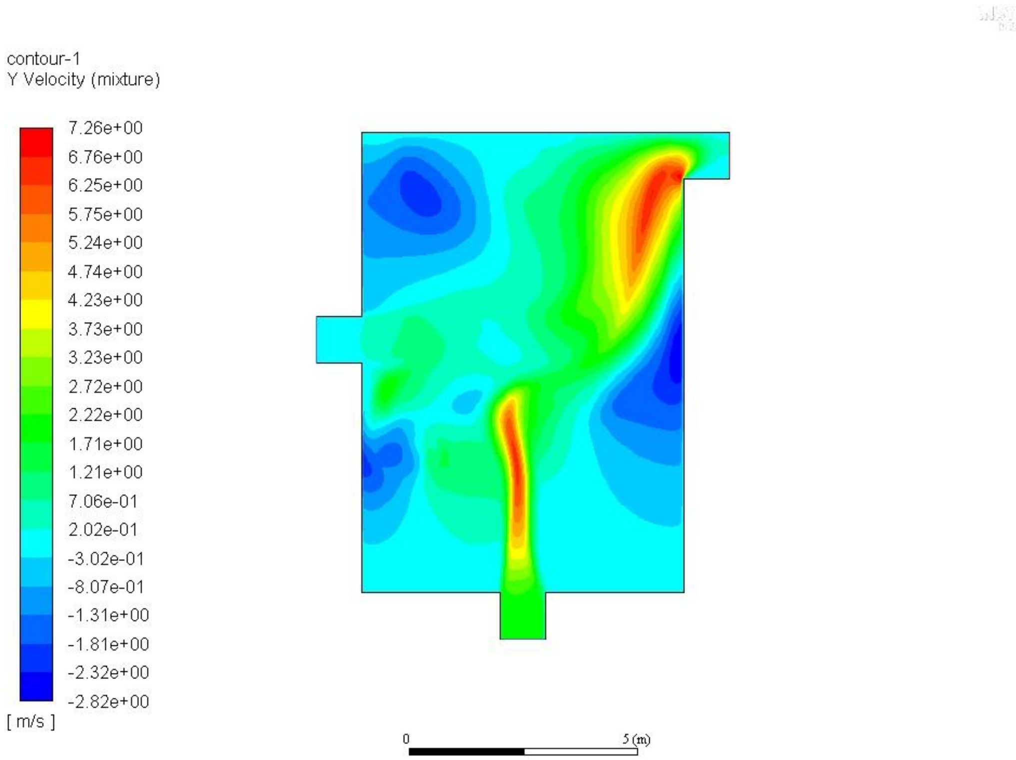
(b) Volume Fraction of methane at t=5s



(c) Contour Plot of X-velocity at t=5s



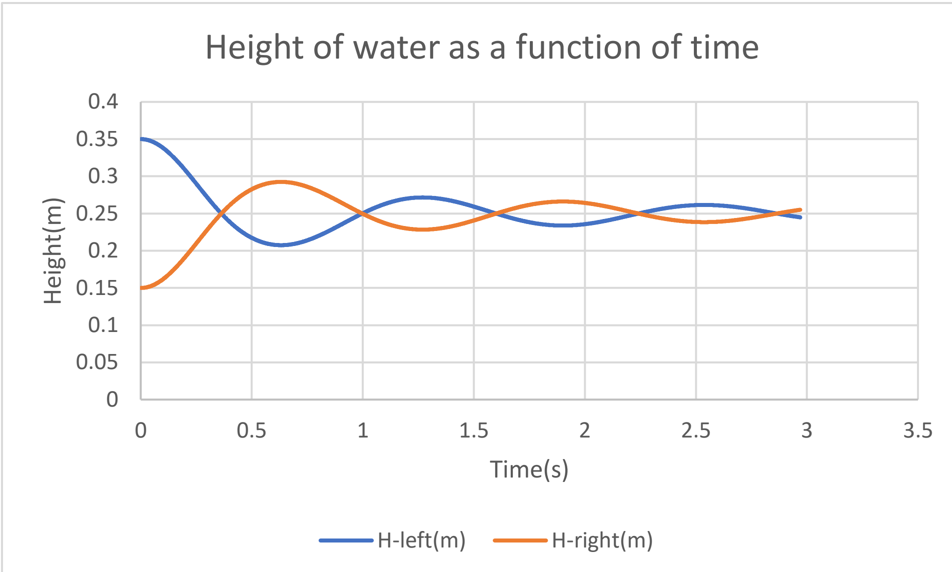
(d) Contour Plot of Y-Velocity at t=5s



3) Discussion: The plot at t=5s show that for higher velocity of air the methane is forced towards outlet unlike the case in task-a where the methane is moving straight up.

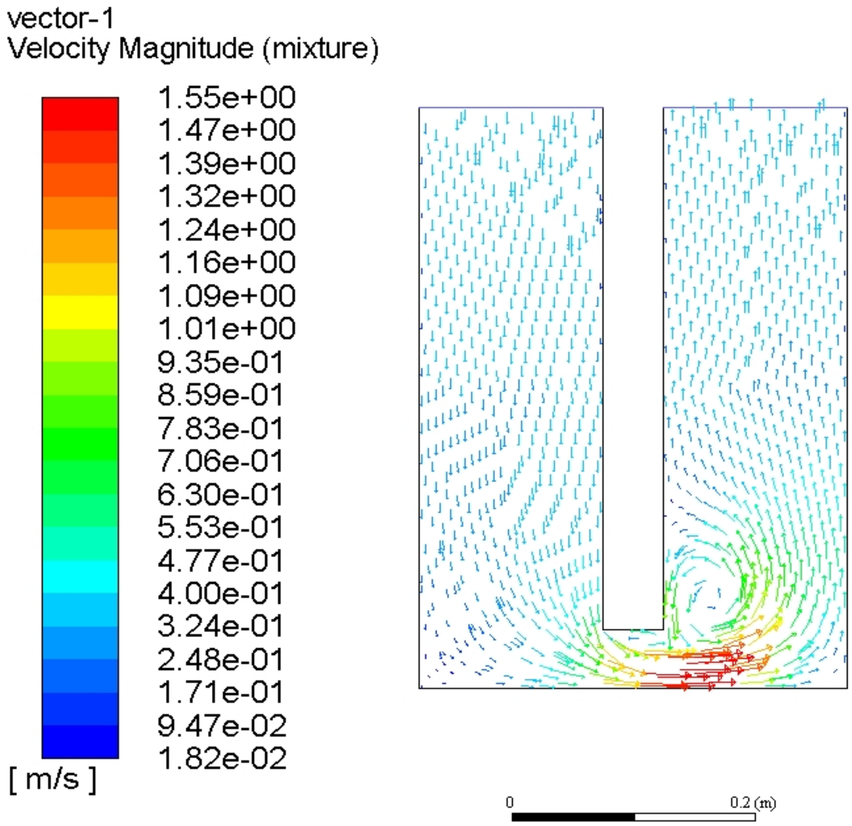
- Task-3-a
  1) Mesh Data: 50,084(nodes)
  2) Solver Setup: Viscous-Laminar, Courant Number: 0.06-0.3, Time step size: 0.001s
  3) Deliverables:
     (a) Boundary Conditions: Pressure Inlets at both open ends(this was chosen by eliminating the other possible choices as outflow cannot be chosen as there is dynamic and rapid change in the air entry and exit at boundaries and outflow would give a wrong convergence and the convergence with outflow was slow. Pressure inlet and outlet cannot be chosen as that would pre-determine the direction of flow which is not a requirement in our case. However, Pressure outlet could be chosen but then there was a slight problem of reverse flow criteria.)

(b) Line Plots of height of water as a function of time

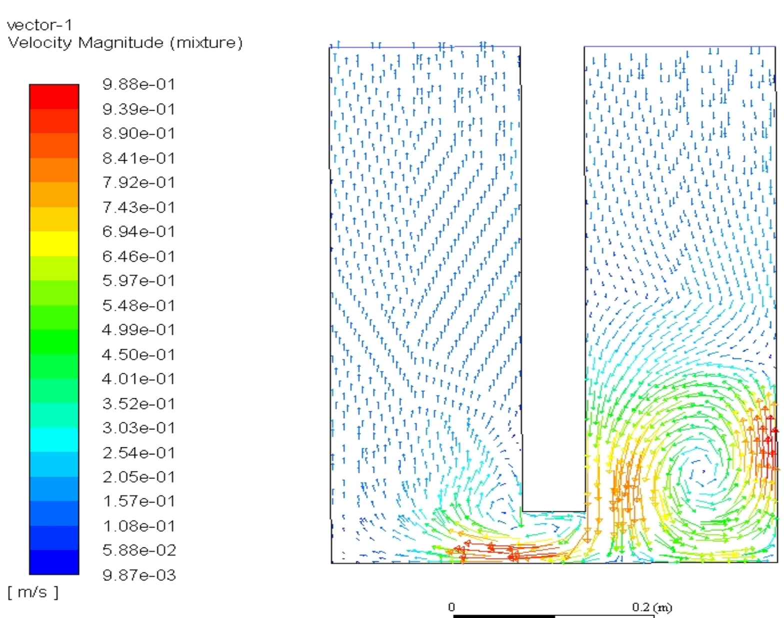


a. Time Period of Oscillation: t1=0.365s, t2=0.965s
b. Period of Oscillation: 1.21s

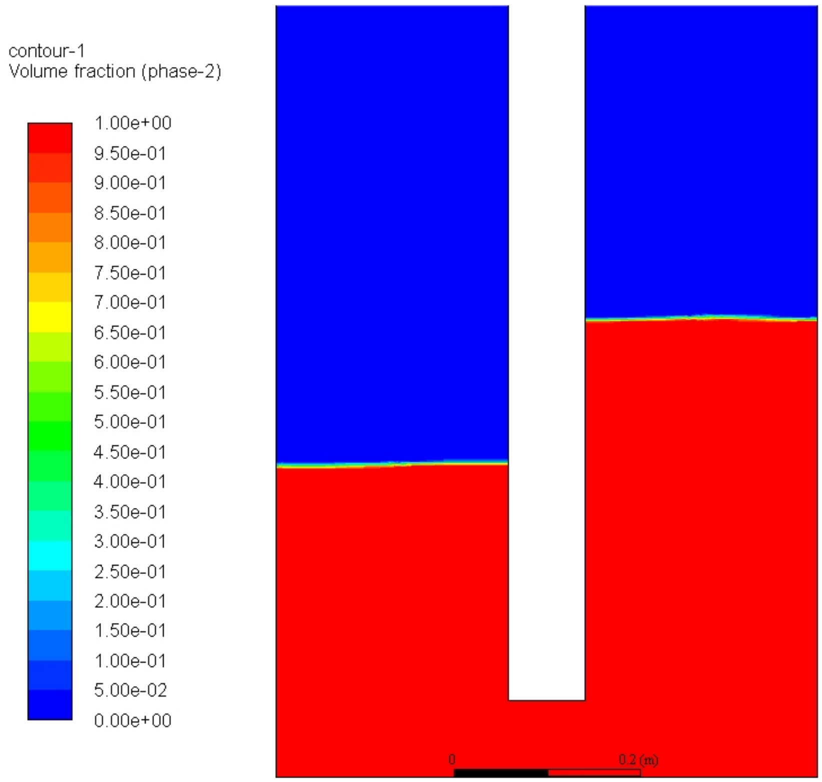
c. Vector plots at t=0.365s



d. Vector plot at t=0.965s

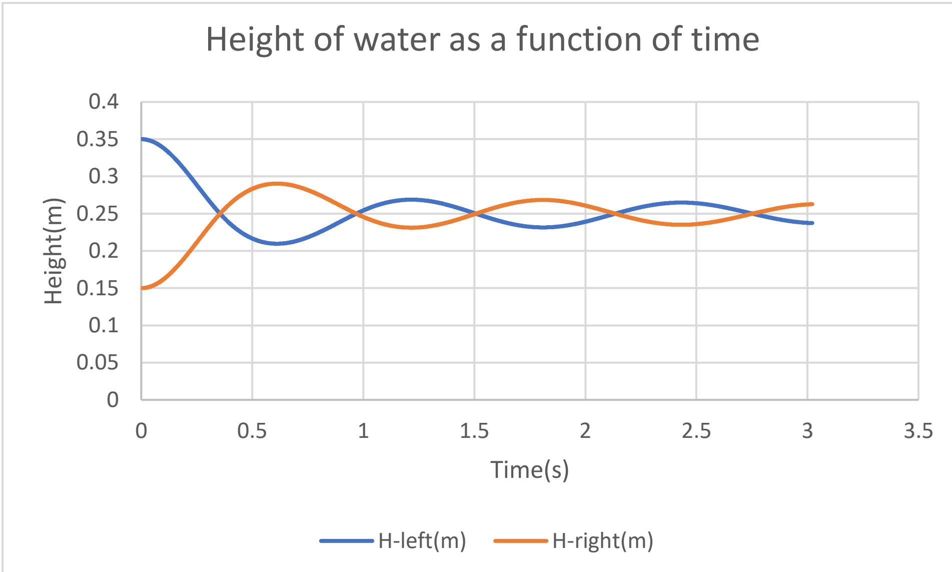


(c) Contour Plot of volume fraction of water when it peaks for first time on right side

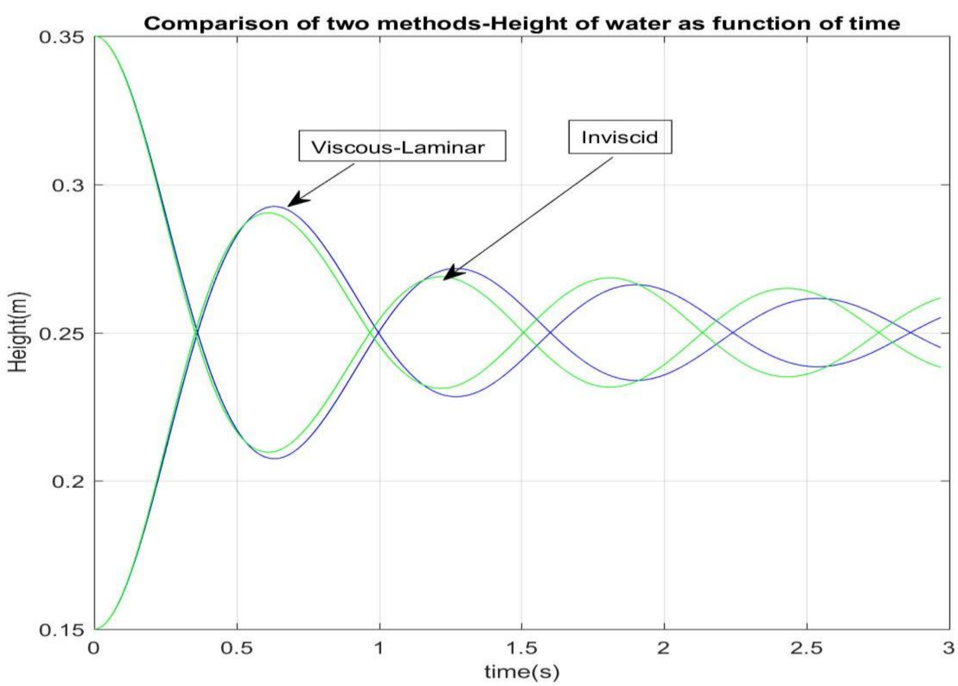


- Task-3-b
  1) Solver Settings: Inviscid Model for Turbulence, Courant Number: 0.06-0.3, Time step size: 0.001s
  2) Deliverables:
     (a) Boundary Conditions: Pressure Inlet at both open ends (this was chosen by eliminating the other possible choices as outflow cannot be chosen as there is dynamic and rapid change in the air entry and exit at boundaries and outflow would give a wrong convergence and the convergence with outflow was slow. Pressure inlet and outlet cannot be chosen as that would pre-determine the direction of flow which is not a requirement in our case. However, Pressure outlet could be chosen but then there was a slight problem of reverse flow criteria.)

(b) Line plots of height of water as function of time



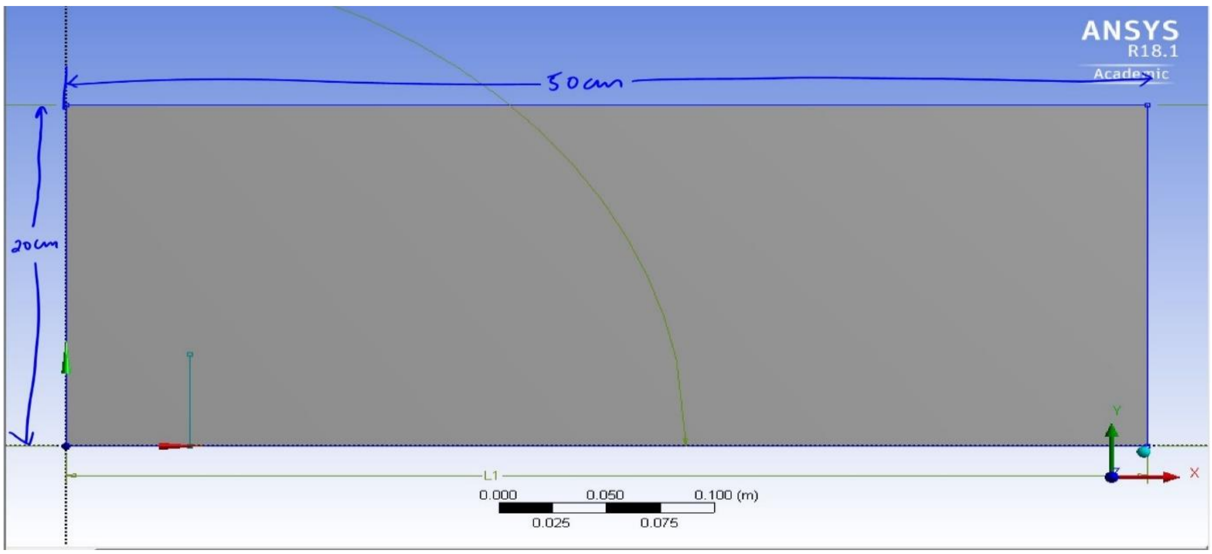
(c) Period of oscillation: 1.19s, t1=0.363s,t2=0.94s



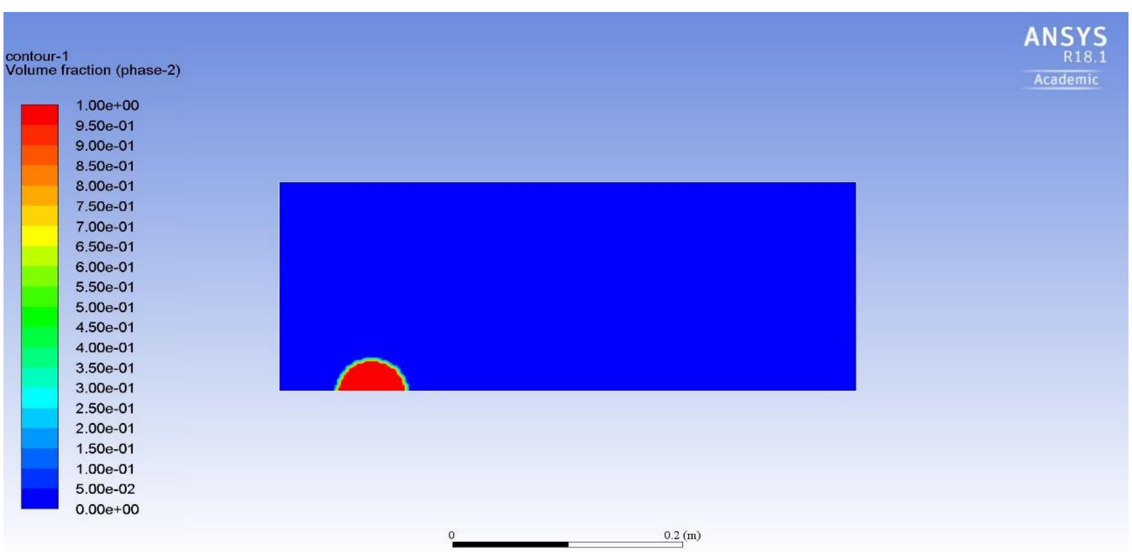
- Discussion:
  From the plot above the oscillations of water that is the difference between the heights of left and right side of water is more for inviscid than viscous laminar due to damping effect by viscosity in viscous laminar model. This difference can be clearly seen at the t=3s. Also inviscid model takes slightly less time for period of oscillations and to reach equal heights as the flow is more turbulent on both sides compared to viscous laminar where the flow needs to move

against friction. However, with the given setup the difference between both the plots in terms of period of oscillation and time to reach equal heights is almost same which is reflected in the first loop where bot overlap.

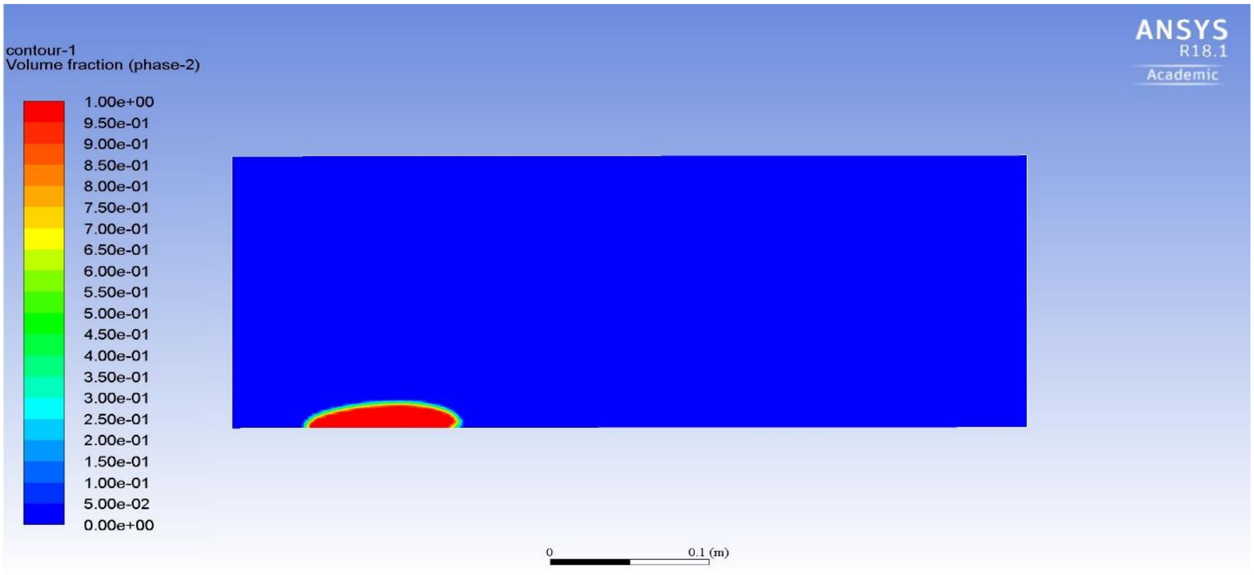
- Task-4-a
    1) Mesh Data: 46,798(nodes)
    2) Solver Setup: Viscous-Laminar, Multiphase Model with air, engine oil, ISO-Surface of 0.9 was used to plot the oil in contours, Courant Number: 0.8-1.2, Time step size: 0.01s
    3) Deliverables:
        (a) Domain Dimensions: L=50cm, H=20cm.



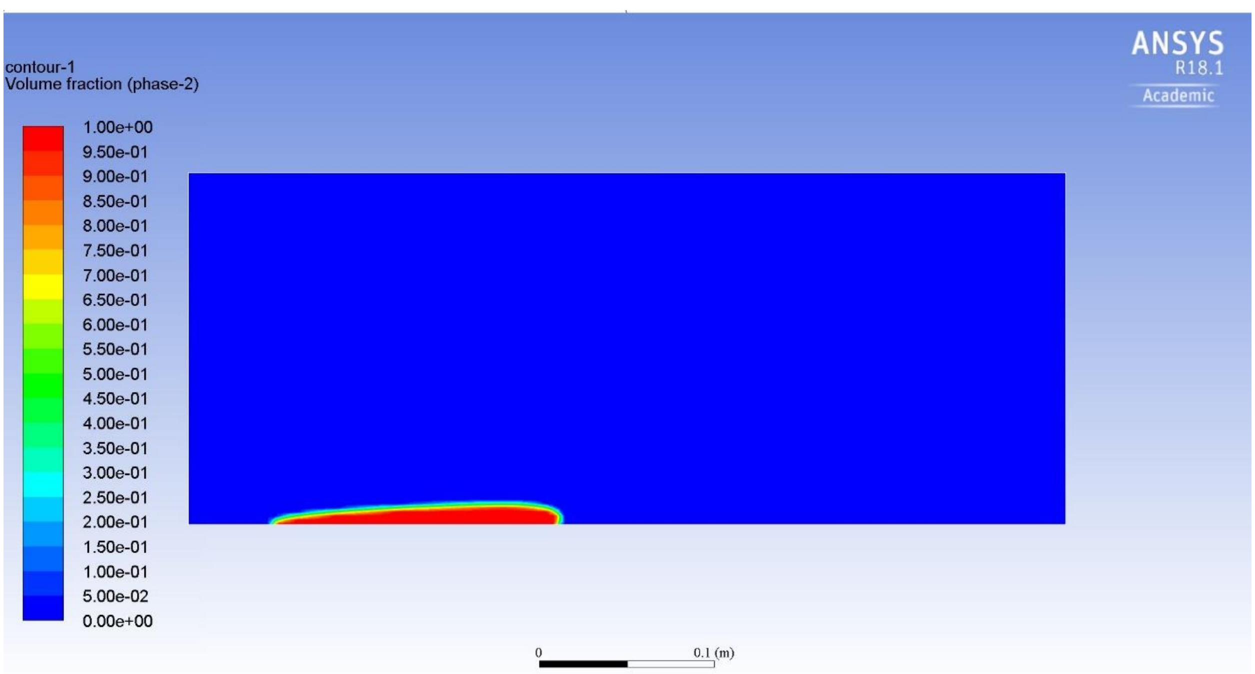
        (b) Mesh: Proximity and Curvature with Refinement of 2- 46,798(nodes), extra inflation layers of 10 at bottom surface.
        (c) Boundary Conditions: Outflow on all sides and the bottom surface was wall
        (d) Contour Plot of Volume Fraction of oil at t=0s



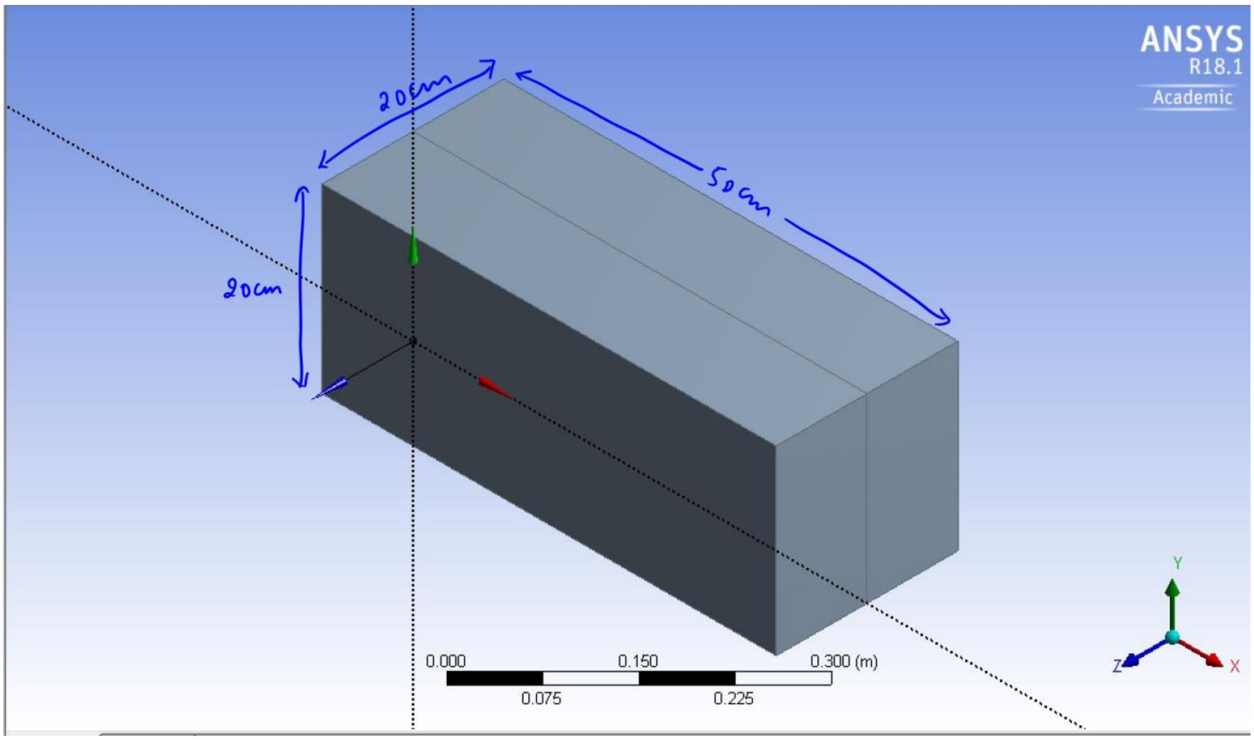
(e) Contour plot of Volume Fraction of water at t=0.1s



(f) Contour plot of Volume Fraction of water at t=0.3s

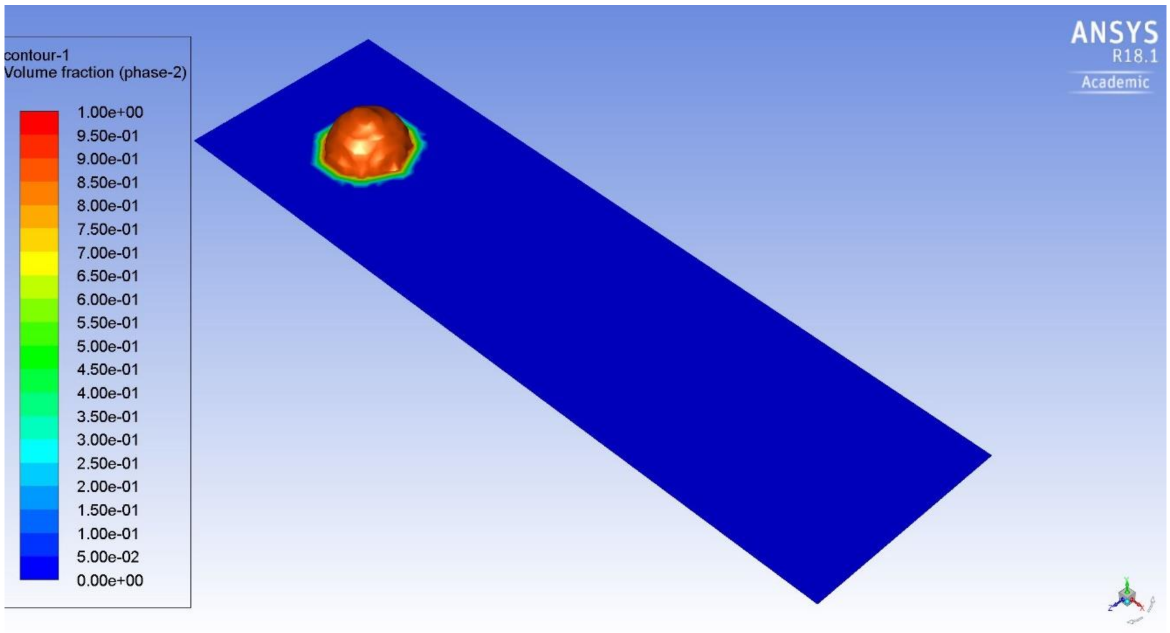


- Task-4-b
  1) Mesh Data: 74,121(nodes)
  2) Solver Data: Viscous-Laminar, Multiphase Model with air, engine oil, ISO-surface of 0.9 was used to plot oil on contours, Courant Number: 0.6-1.3, Time step size: 0.005s
  3) Deliverables:
     (a) Domain Dimensions: Length=50cm, Height=20cm, Depth=20cm (such that the oil center will be at center of that dimension i.e., z=0)

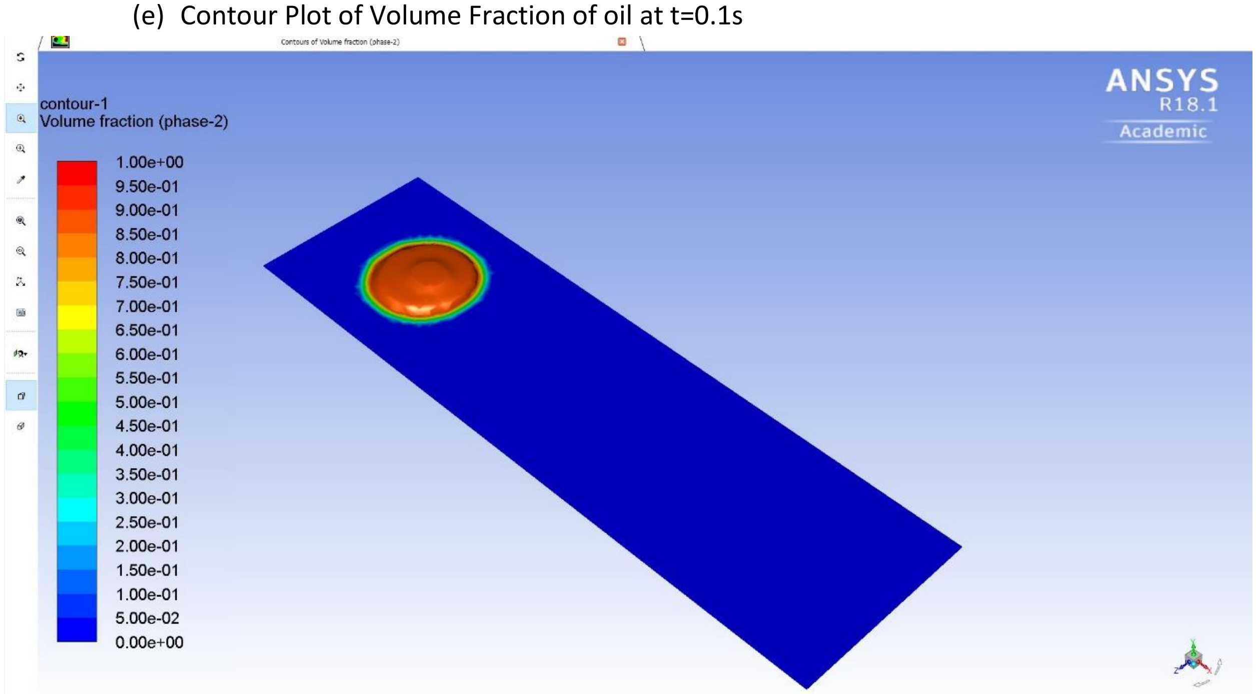


     (b) Mesh Setup: Proximity and Curvature with Refinement of 2(74121 nodes), also element size of 0.009m was used to further refine the mesh with extra inflation layers(20) on the bottom surface.
     (c) Boundary Conditions: Outflow on all sides and the bottom surface was wall

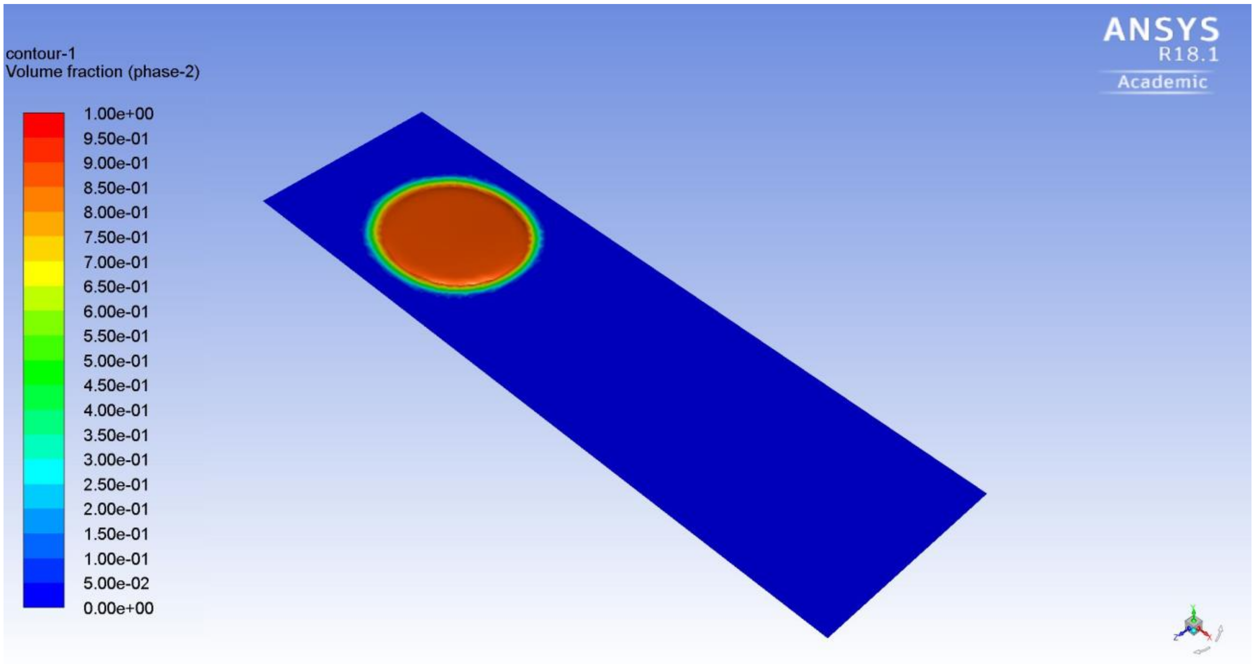
(d) Contour Plot of Volume Fraction of oil at t=0s



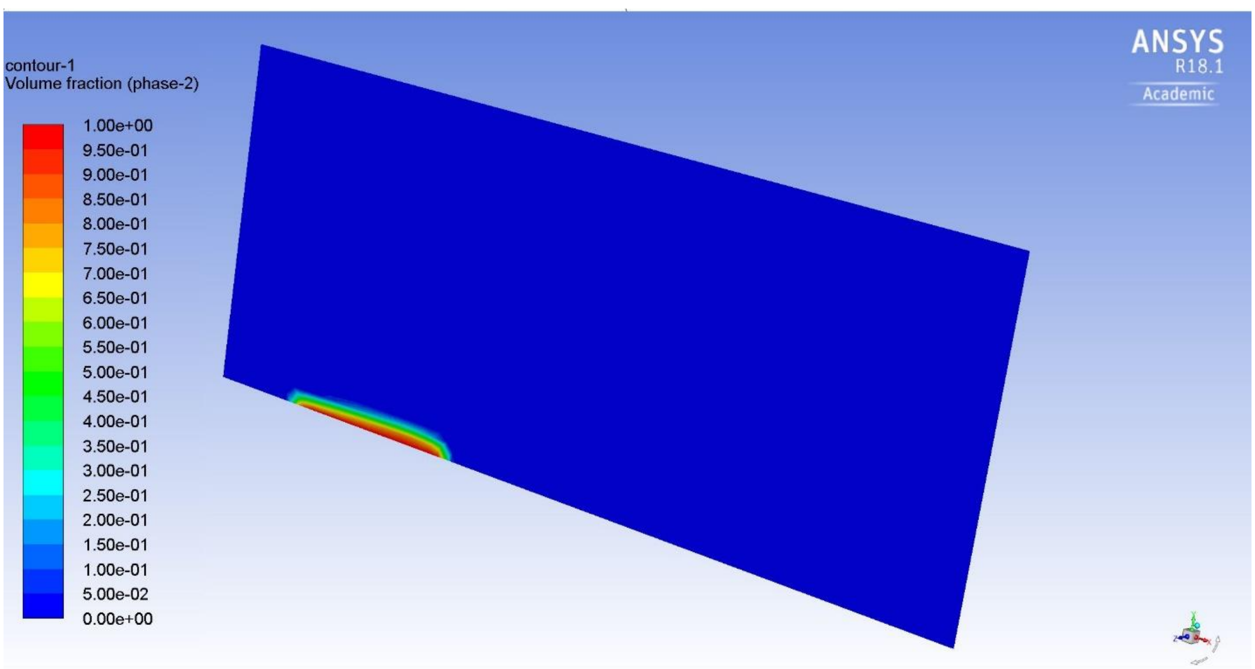
(e) Contour Plot of Volume Fraction of oil at t=0.1s



(f) Contour Plot of Volume Fraction of oil at t=0.3s



(g) Side View of Contour Plot of Volume Fraction of oil at t=0.3



- Discussion of task-a&b: the mesh setting was chosen based on mesh refinement test where nodes ranging from 30,000 to 70,000 were tested to see their impact on the result. Near the bottom surface special care is taken to add extra inflation layers for better estimation of boundary layer effects. Also outflow is chosen since there is no pressure or high velocity involved in terms of movement of air in the domain. Besides this particular setting helped in a faster convergence compared to others.