Numerical simulations of impact of a droplet on the surface of water using OpenFOAM

Dr. Raj Kumar Saini

Ph.D, Indian Institute of Technology, Bombay (IIT Bombay)

M.Tech, Indian Institute of Technology, Madras (IIT Madras)

Email : raj.km.saini@gmail.com

January 4, 2020

Abstract

This case study demonstrates the simulation of the impact of a droplet on the surface of water pool. The study of the impact of a Newtonian (water) droplet on the surface of water pool is interesting to understand the droplet hydrodynamics. The numerical simulations for tracking the free surface motion are carried out using volume of fluid (VOF) method. Impacting of a Newtonian droplet on a water pool investigated numerically. The planer geometry (2D) and meshing of the domain are created using SALOME-9.3.0. Radius of the droplet is defined and sets in the setFieldsDict. The simulations are performed using OpenFOAM-v6. The simulations results are compared and analyzed of two cases with and without adaptive mesh refinement.

Keywords: Hydrodynamics, Droplet deformation, Volume of Fluid, Two-phase flow, CFD

1 Introduction

The purpose of this case study is to learn OpenFOAM software [1] to the new users. The impact of a water droplet on the surface of water in the cavity is interesting to understand the hydrodynamics of droplet like drop deformation. This case demonstrates and implementation of mesh refinement during simulation when the droplet shape continuous changing [2, 3, 4]. In the simulations, a pressure-based finite volume method is used for incompressible flow. The simulations are performed using OpenFOAM-v6 [1]. In this case, two phases flow simulation approaches are considered for incompressible transient flow with using **interFoam** solver in OpenFOAMO-v6 [1]. This case study demonstrates how to do the following:

- Set up a problem case;
- Creating a 2D mesh by using Salome (mesh.unv);
- Mesh imported in to OpenFOAM (ideasUnvToFoam);
- solve a transient problem using the VOF model;
- Create the geometry and import the geometry in OpenFOAM;
- Set up the properties of the fluids;
- Initialize the flow conditions;
- Set adaptive mesh refinement for the interface;
- Consider transient and laminar flow regime;

- Set boundary/initial conditions (BC/IC);
- Brief explanation of mesh refinement;
- Post processing the case for results.

2 Problem statement

Numerical simulations are performed to study the hydrodynamics of water droplet in a cavity. Figure 1 shows the details of the geometry considered in the present study. This case explains setting up OpenFOAM case in the OpenFOAM repository. The geometric parameters of the domain such as height, width and depth are considered with 400x400x1 (units, 40 mm x 40 mm x 10 mm) respectively. Table 1 shows the fluid properties for the present study.

The radius of droplet is defined with setfieldsDict in a 2D environment of the continuous phase(air). The dispersed phase is considered water. Initially, the domain is filled with air and the bottom is filled with water up to a certain height (5 mm). The disperse phase (water) is patched in a position (0.02 0.038 0) with radius size of 2 mm.

At time zero, the water droplets are patched and will start moving to downwards with velocity. Δt and write intervals are 0.00001 s and 0.002 s considered respectively.

The velocity of the water droplets is 2 m/s (-ve, y-direction). The time needed to stabilize the system is about t sec.

Table 1: Details of fluids property

Dynamic viscosity, water (μ_1), Pa.s	1e-06
Dynamic viscosity, air (μ_2), Pa.s	1.48e-05
Density, water (ρ_1), kg/m^3	1000
Density, air (ρ_2), kg/m^3	1
Surface tension, water-air (σ), N.m	0.07

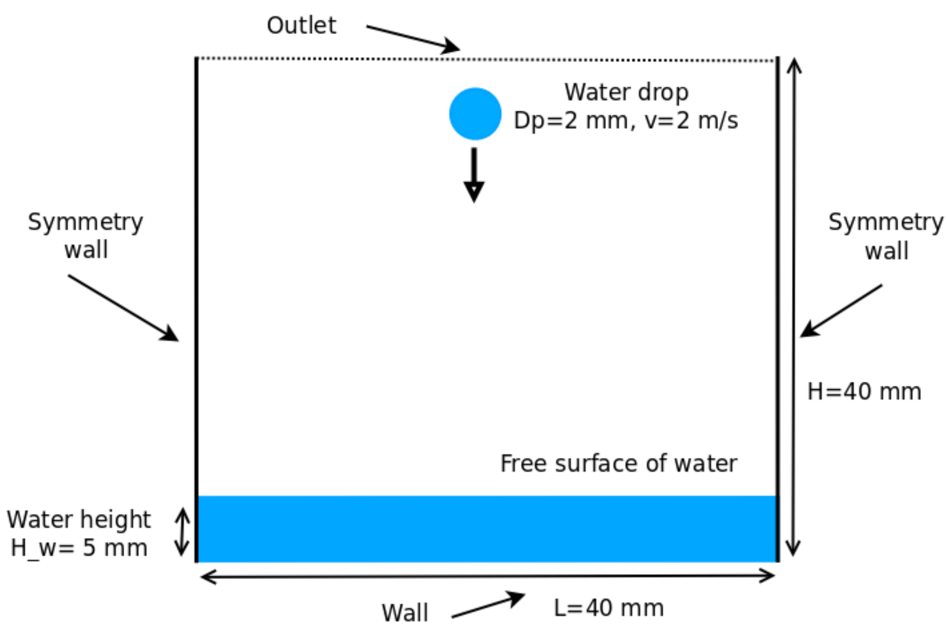


Figure 1: Schematic diagram

3 Mathematical modeling

InterFoam (solver) solves the Navier-Stokes equations for isothermal and incompressible flow. It means that all material properties are constant in the region filled by one of the two fluid except at the interphase.

3.1 Continuity equation

The continuity equation (constant-density) is defined as:

$$\frac{\partial u_j}{\partial x_j} = 0 \tag{1}$$

3.2 Momentum equation

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\tau_{ij} + \tau_{t_{ij}}\right) + \rho g_i + f_{\sigma i} \tag{2}$$

where, u represents the velocity, ρ density of the fluid, p the pressure, g_i the gravity and $\tau_{t_{ij}}$, τ_{ij} and $f_{\sigma i}$ are the turbulent, viscose stresses and the surface tension respectively.

The density of mixture fluid (ρ) is defined as (Equation- 3):

$$\rho = \alpha\rho_1 + (1-\alpha)\rho_2 \tag{3}$$

The value of α is 1, it means that the fluid 1 occupied with the density ρ_1 and the fluid 2 is occupied with the density ρ_2 with 0 the value of α. The α varies between 0 and 1 at the interphase between the two fluids.

The volume fraction α is specified as (Equation- 4):

$$\alpha = \begin{cases} 1 & \text{water phase} \\ 0 < \alpha < 1 & \text{air - water interface} \\ 0 & \text{air phase} \end{cases} \tag{4}$$

The Continuum Surface Force (CSF) approach is used to modelled the surface tension ($f_{\sigma i}$) [5, 6]. It can be calculated using Equation- 5:

$$f_{\sigma i} = \sigma\kappa\frac{\partial \alpha}{\partial x_i} \tag{5}$$

σ and κ are the surface tension constant and radius of the curvature respectively. The curvature can be estimated using Equation- 6 [5, 6].

$$\kappa = -\frac{\partial n_i}{\partial x_i} = -\frac{\partial}{\partial x_i}\left(\frac{\partial\alpha/\partial x_i}{|\partial\alpha/\partial x_i|}\right) \tag{6}$$

3.3 Volume of fluid

In order to investigate where the interphase between the two fluids (water, air) is, an additional equation for α is to be solved.

$$\frac{\partial \alpha}{\partial t} + \frac{\partial(\alpha u_j)}{\partial x_j} = 0 \tag{7}$$

3.4 Boundary conditions for fields variables

Details of boundary name and corresponding boundary conditions are shown in Table 2.

Table 2: Boundary conditions

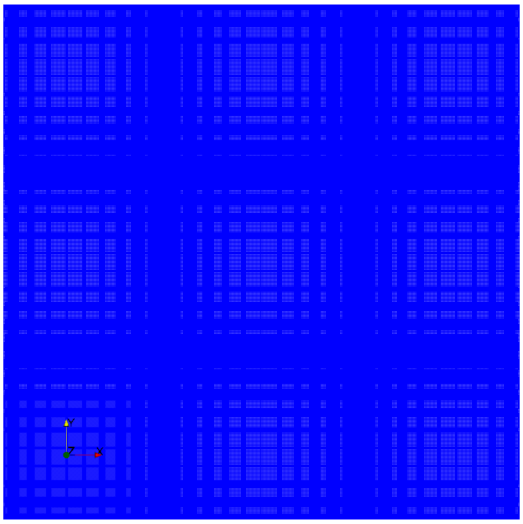
Boundary Name	Boundary condition
Symmetry1	symmetryPlane
symmetry2	symmetryPlane
Wall	No slip
Outlet	Pressure
FrontAndBack	empty

4 Simulation procedure

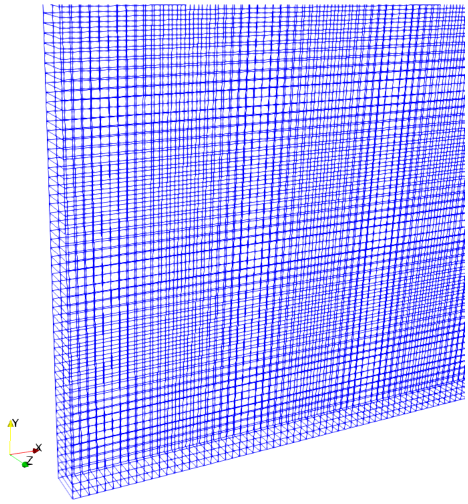
This case deals with two-dimensional laminar simulation. First step in setting up of an OpenFOAM case is to copy to present working directory. We need to set all require input parameters before starting the simulation. Mesh generation and implementation of boundary conditions are adopted from a base damBreak tutorial. This case study is considered with transient and laminar flow. The solver settings are imported from damBreak tutorial (OpenFOAM/(username)-6/run/tutorials/multiphase/interFoam/laminar/damBreak/damBreak/) from **interFoam**.

4.1 Creating geometry and mesh

- Geometry for the present problem is considered 3D dimensional domain. The geometry and mesh are generated by using the Salome, the file (mesh.unv) is available in 'case/cad' folder in user directory (∼/case/cad).
- Mesh can be imported to in OpenFOAM format by using the command 'ideasUnvToFoam' in the terminal.
- Figure 2 shows the isometric view of the generated mesh using ideasUnvToFoam utility.
- All modifications for mesh with proper boundary condition with defining inlet as velocity patch and its neighbor patch (∼/case/constant/polyMesh/boundary).



(a) Mesh



(b) Zoom view

Figure 2: Computational geometry

4.2 Setting boundary conditions (BC)/Initial conditions (IC)

All the boundary conditions for fields variables are mentioned in '0' file folder. Files present in '0' folder (∼/case/0) has been kept 'alpha.water', 'p_rgh' and 'U' files for laminar and transient flow. Boundaries are assigned and added six boundaries of present case in three the files, i.e. 'alpha.water', 'p_rgh' and 'U' respectively. Details of the boundary conditions are listed in Table 3.

Table 3: Details of boundary conditions for fields variables

Boundary	p_rgh	U	alpha.water
symm1	symmetryPlane	symmetryPlane	symmetryPlane
symm2	symmetryPlane	symmetryPlane	symmetryPlane
inlet (assigned as wall)	fixedFluxPressure	fixedValue	zeroGradient
outlet	totalPressure	pressureInletOutletVelocity	inletOutlet
frontAndBack	empty	symmetryPlane	empty

4.3 Dynamic Mesh Refinement for 2-dimensional

Standard OpenFOAM has a module called dynamicRefineFvMesh. The solver can do the refinement only for the 3D hexahedral cells by partitioning the cells equally in all three directions. This adaptive mesh refinement tool does not seem to be applicable to 2D-dimensional problems. The modified code to adapt this adaptive mesh tools for two-dimensional problems [2, 4]. The working code has shared for adaptive mesh refinement for 2d problems. In this study, we concentrate on the dynamic mesh refinement for 2-dimensional. The mesh refinement is achieved by using the **dynamicRefine2DFvMesh** and **dynamicMesh** libraries. For more details for dynamic mesh refinement, please refer open form used the guide and cfd-online forum [3].

Compilation of adaptive mesh refinement, Switch to **src** directory and then run : **wmake libso**.

```
src
|----dynamicRefine2DFvMesh/
```

```
        |----dynamicRefine2DFvMesh.C
        |----dynamicRefine2DFvMesh.H
        |----U
|----hexRef2D/
        |----hexRef2D.C
        |----hexRef2D.H
|----Make/
        |----files
        |----options
```

Here is the brief introduction of **OpenFOAM**, a toolbox of CFD simulation: "OpenFOAM" is an open source toolbox for CFD simulations. "interFoam" is an open source CFD solver of OpenFOAM. The structure of the **case** is like this:

```
case
|----0/
        |----alpha.water
        |----p_rgh
        |----U
|----system/
        |----polyMesh/
        |----dynamicMeshDict
        |----g
        |----transportProperties
        |----turbulenceProperties
|----constant/
        |----changeDictionaryDict
        |----controlDict
        |----fvSchemes
        |----fvSolution
        |----setFieldsDict
|----clean.sh
|----run.sh
```

4.4 Solver details

The volume of fluid (VOF) method for phase fraction is used for incompressible, isothermal, immiscible fluids based on interface capturing approach with optional mesh and geometrical reconstruction technique is used (piecewise linear interface calculation) in simulation. In the present study, unsteady state and laminar flow are considered. Turbulent flow model can be applied in the OpenFOAM by using 'simulationType' option in the 'turbulenceProperties' file in constant folder. The 'simulationType' option can be used as '**laminar**'. In order to run transient simulations, controlDict, decomposeParDict (for parallel computation), fvSchemes,fvSolution, and setFieldsDict files are kept in the system directory folder. '**./run**' command executes in terminal to run computations.

4.5 Post-processing

The paraFoam, it can be used to visualize the simulations results in OpenFOAM. This can be run by typing the following command line in the terminal **paraFoam** to open the ParaView software and upload the case.

5 Results and discussion

Simulations are performed using OpenFOAM to investigate the hydrodynamics of the droplets. Simulations results are plotted with the help of 'paraview' ($\sim$/case/paraFoam). Simulation results are analyzed with the help of paraview software.

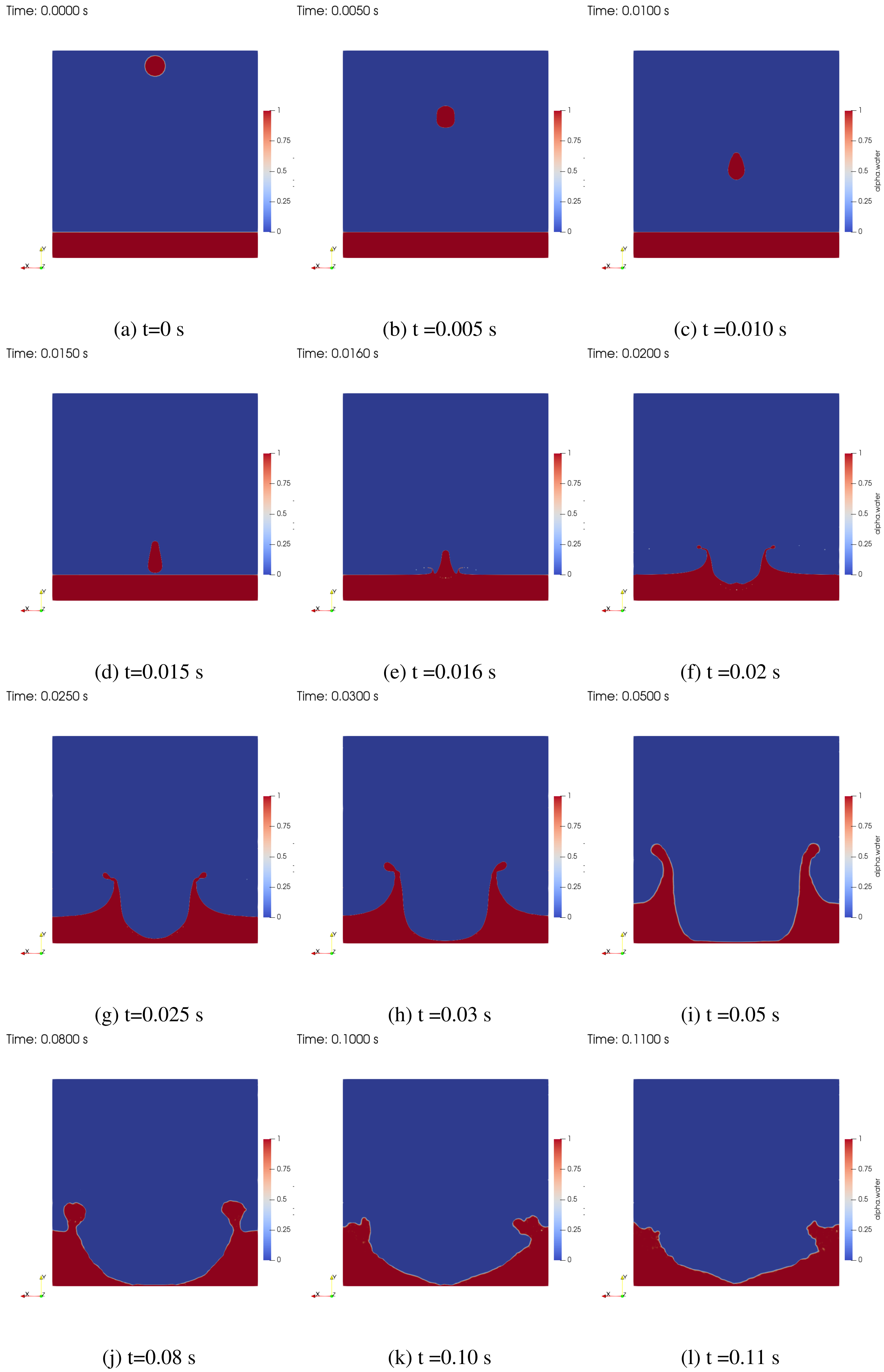
In this section, the numerical results for water droplet impacting on surface of water pool are presented. The physical properties of fluids are given in Table 1.

Figure 3 shows successive stages of droplet deformation for a different time, viewed along the z-axis. A 2 mm diameter water droplet impacts the surface of the water pool, with an impact velocity of 2 m/s. The flow field such as volume fraction of water contour shows in Figure 3. At time 0.001, the refined mesh can be seen at the interface in Figure 3a, which is not seen in Figure 3b.

Figure 3d at time 0.015 sec, then the droplets can be visualized before the hitting surface of the water at the bottom. It can be seen in Figure 3d while the first droplet hits the water surface. It is observed that similar, simulations results are shown for different times.

Figure 3 l does not show the settling down of waves at the bottom. As time passes, the system will start to settle at any moment of time. In this work, the simulation can start further and will obtain the extract time value when the wave completely settles down or system archived steady state.

Figures 4 & 5 show sequential images of the impact of a 2 mm diameter water droplet at 2 m/s. The simulation result shows the comparison of the field α (volume fraction) for water droplet with and without mesh refinement at the interface for different times. It is observed that the simulations results with mesh refinement are capture more realistic dynamic of the droplet (like shape, deformation of the droplet) as shown in Figures 4a, c & e and Figures 5a, c & e. It can be explained similarly for different times, which are obtained from the simulations in both cases.



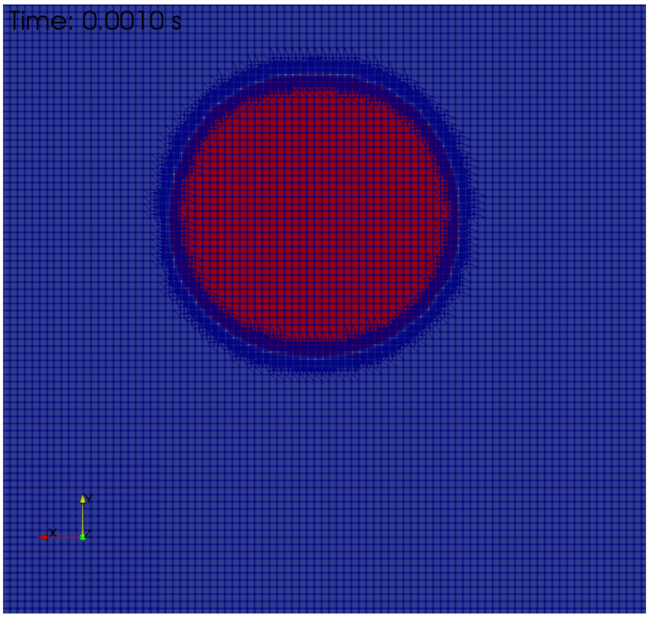
(a) t=0 s (b) t =0.005 s (c) t =0.010 s

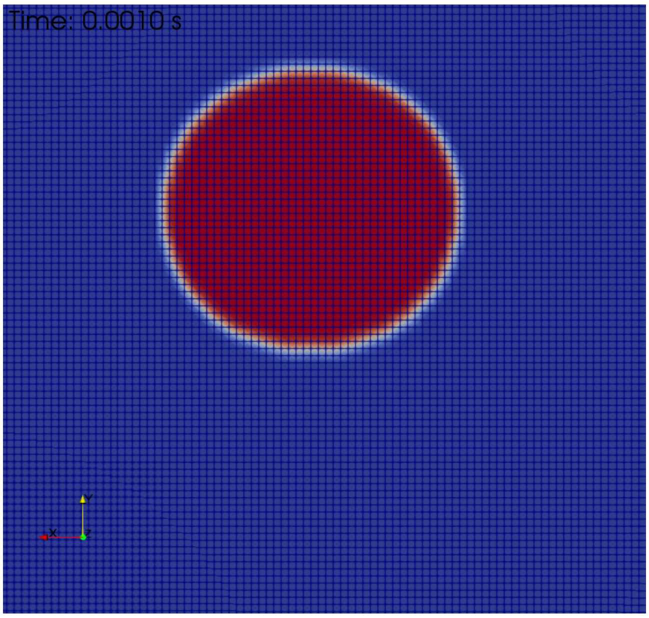
(d) t=0.015 s (e) t =0.016 s (f) t =0.02 s

(g) t=0.025 s (h) t =0.03 s (i) t =0.05 s

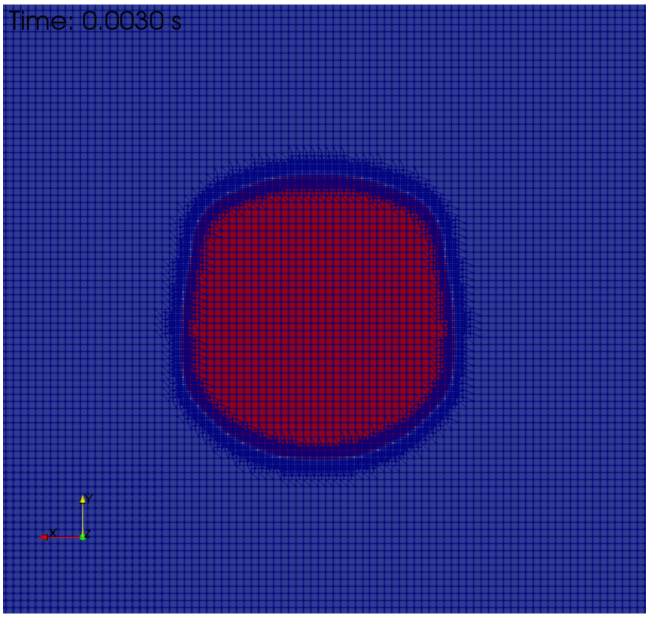
(j) t=0.08 s (k) t =0.10 s (l) t =0.11 s

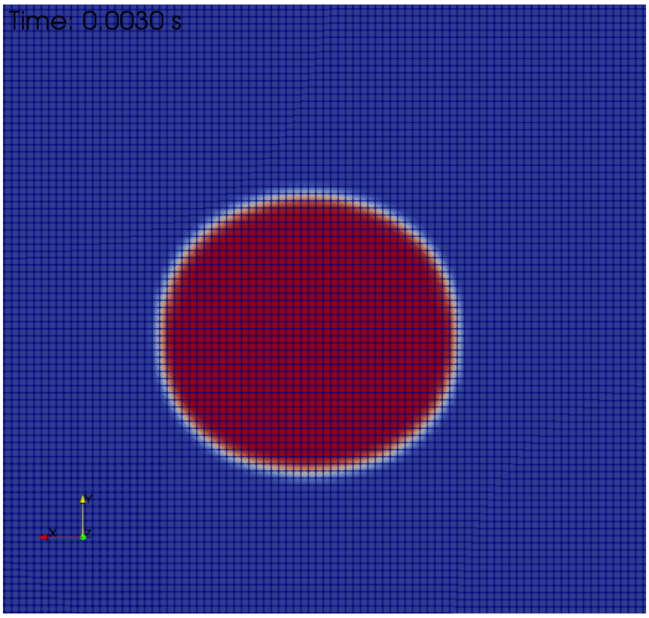
Figure 3: Field α or volume fraction for different time (red color correspond to α=1, water and blue color corresponds to α= 0, air) with droplet velocity, v=-2 m/s, (mesh refinement).



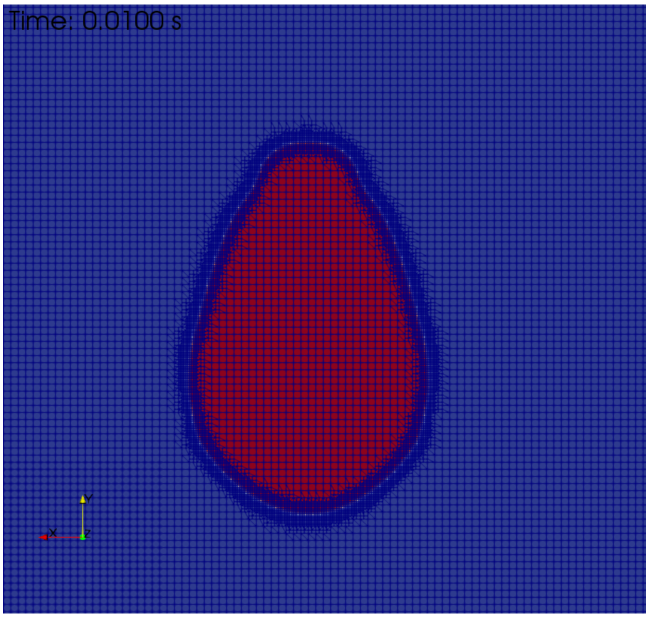


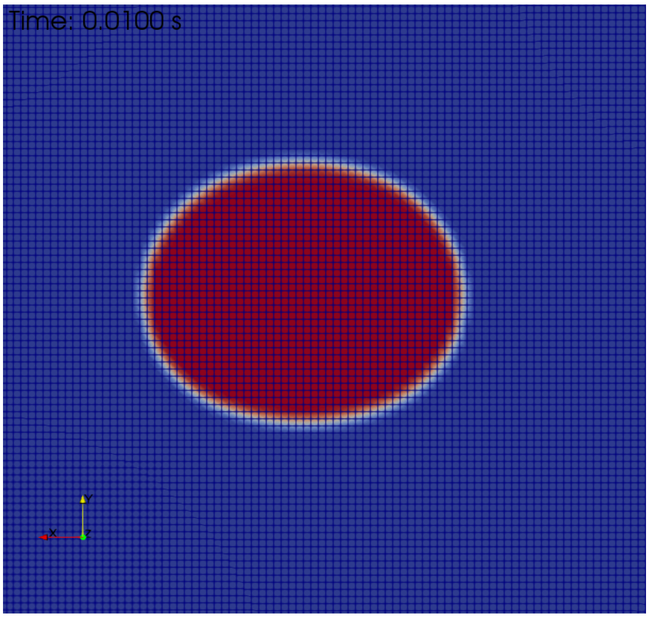
(a) t =0.001 s (with mesh refinement) (b) t =0.001 s (without mesh refinement)





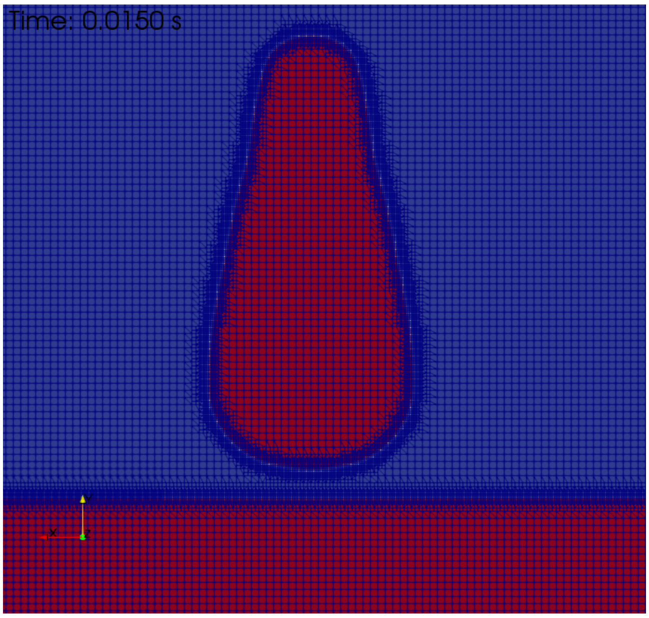
(c) t =0.003 s (with mesh refinement) (d) t =0.003 s (without mesh refinement)



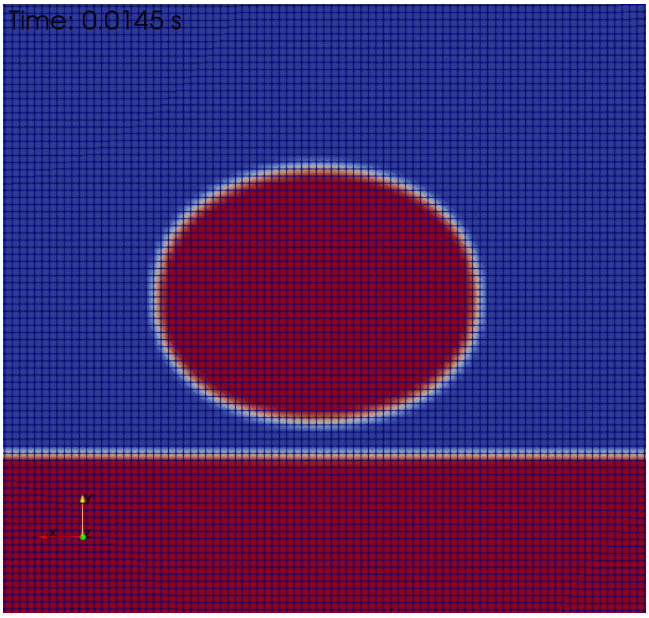


(e) t =0.01 s (with mesh refinement) (f) t =0.01 s (without mesh refinement)

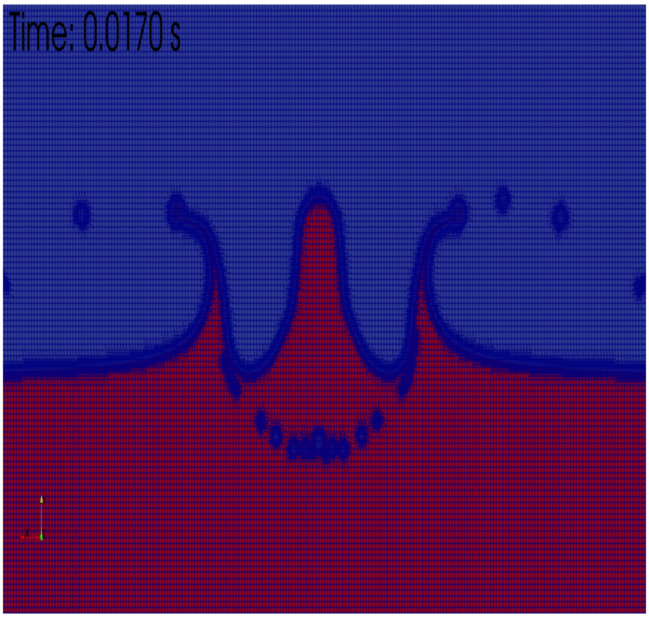
Figure 4: Field α or volume fraction for different time with droplet velocity, v=-2 m/s (red color correspond to α=1, water and blue color corresponds to α= 0, air), with and without mesh refinement.



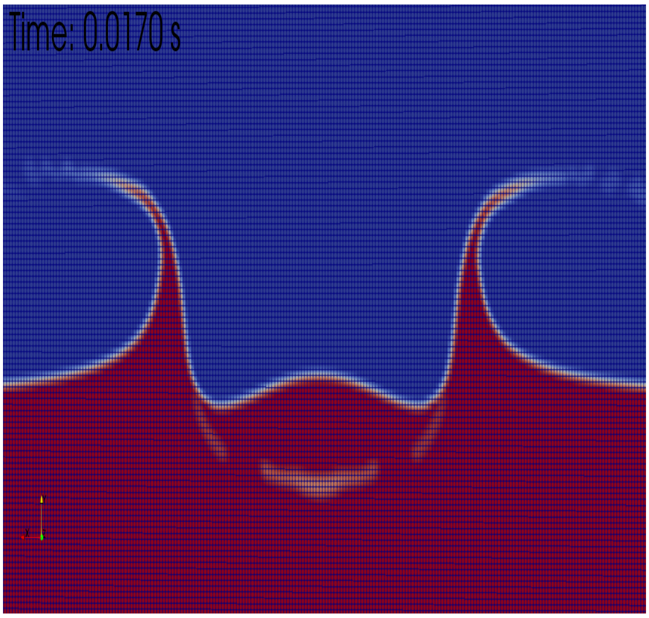
(a) t =0.015 s (with mesh refinement)



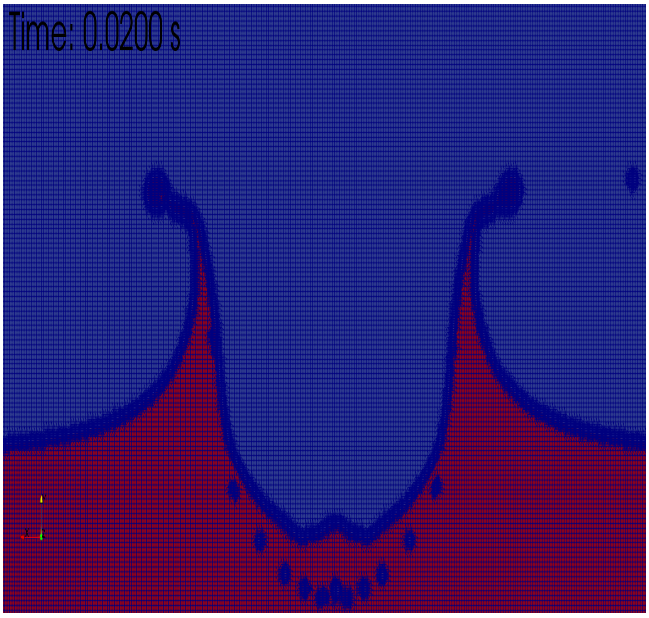
(b) t =0.015 s (without mesh refinement)



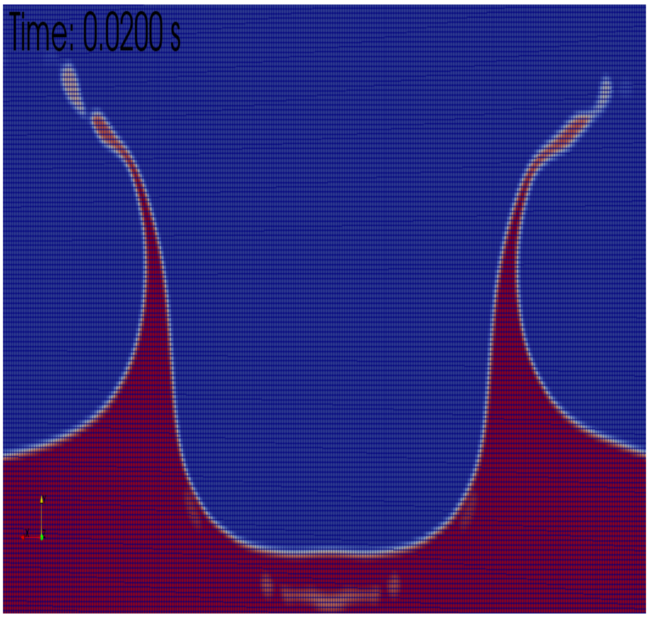
(c) t =0.0017 s (with mesh refinement)



(d) t =0.0017 s (without mesh refinement)



(e) t =0.02 s (with mesh refinement)



(f) t =0.02 s (without mesh refinement)

Figure 5: Field α or volume fraction for different time with droplet velocity, v=-2 m/s (red color correspond to α=1, water and blue color corresponds to α= 0, air), with and without mesh refinement.

References

[1] C. J. Greenshields, *OpenFOAM: The OpenFOAM Foundation. User Guide Version 6*. CFD Direct Limited, July. 2018.

[2] A. Baniabedalruhman, *Dynamic meshing around fluid-fluid interfaces with applications to droplet tracking in contraction geometries*. Michigan Technological University, 2015.

[3] C. O. D. Forums. Retrieved from. [Online]. Available: https://www.cfd-online.com/Forums/openfoam-community-contributions/118870-2d-adaptive-mesh-refi

[4] Academia. Retrieved from. [Online]. Available: https://www.academia.edu/16217705/Two-dimensional_adaptive_meshing_in_OpenFOAM

[5] J. Brackbill, D. Kothe, and C. Zemach, "A continuum method for modeling surface tension," *Journal of Computational Physics*, vol. 100, no. 2, pp. 335 – 354, 1992.

[6] J. A. Heyns and O. F. Oxtoby, "Modelling surface tension dominated multiphase flows using the vof approach," 2014.