Numerical simulations of freely falling droplets of water in OpenFOAM and Fluent

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

December 16, 2019

Abstract

This case study demonstrates the simulation of freely falling of water droplets in the container like a cavity. This case study is designed such that the properties of the system are similar to those of raindrops falling through the air. The study of multiple droplets is interesting to understand, the phenomena of drop dynamics like drop coalescence and breakage from an engineering point of view. Here, the planer geometry (2D) and meshing of the domain are created using SALOME-9.3.0. The multiple radii of droplets are defined and are set in the setFieldsDict. The simulations are performed using OpenFOAM-v6 and Ansys Fluent-v19.2. The dynamic behavior of the droplets is captured. The simulations results of Openfoam and Ansys-Fluent are compared and analyzed.

Problem Statement

The geometric parameters of the domain such as height, width and depth are considered with 400x400x1 (units, 4 cm x 4 cm x 1 cm) respectively. The radius of droplets (sphere) are defined with setfieldsDict in a 2D environment of the continuous phase (air). The dispersed phase is considered water. Initially, disperse phase (water) is patched in different position with multiple radius of between 0.3 - 2.0 mm respectively.

- Creating a 2D mesh by using Salome (cavity_mesh_square.unv);
- Mesh imported in to OpenFOAM (ideasUnvToFoam);
- Set boundary/initial conditions (BC/IC);
- Solver- **interFoam** .

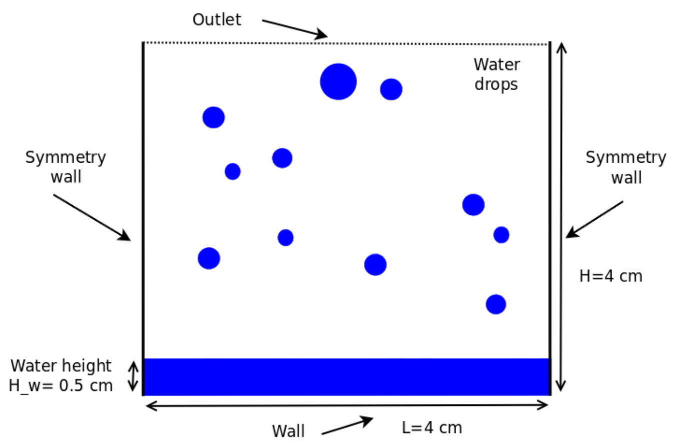

Figure 1: