

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.

Problem Statement

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. The radius of droplet (sphere) is defined with setFieldsDict in a 2D environment of the continuous phase (air). The dispersed phase is considered water. Initially, disperse phase (water drop) is patched with radius of 2 mm.

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

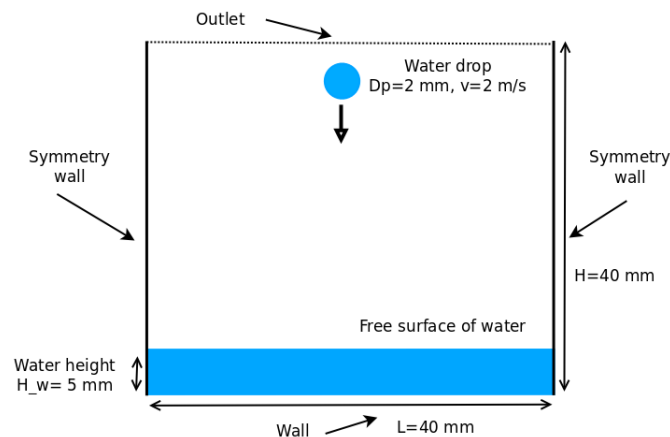


Figure 1: