Numerical simulations of water flow in a open channel 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 10, 2020

Abstract

This case study demonstrates the simulation of water flow in an open channel. This kind of problem has many many applications such as canals, streams, rivers, & sewerage system etc. The open channel flow is also referred to as gravity flow or free-surface flow. In present the study, a horizontal rectangular channel with inclined sides is considered with two side outlets. Numerical simulations are carried out to investigate the flow characteristics in the channel flow. Volume of Fluid (VOF) method approach is used for tracking the free surface of the water. The geometry and meshing of the domain are created using SALOME-9.3.0, blockMesh and snappyHexMesh toots. The simulations are performed using OpenFOAM-v6. The simulation results of velocity profiles and volume fraction are analyzed in channel flow.

Problem statement

Solving incompressible, transient and isothermal flow in an open channel(3D) as shown in Figure 1. Water enters into channel with flow rate (Q, m^3/s). Initially, it is filled with air with very less water at the bottom (up to 0.3 m of height channel) and then water enters the open channel. The geometrical parameters are shown as domain (11 m x 3 m x 1.5 m); two outlets (1 m x 1 m). It's a purpose to describe and dealing with parallel computations, two-phase system dealing with open-source CFD package OpenFOAM.

- Creating background a 3D mesh by using blockMesh utility;
- Creating surface files (stl) using SALOME-9.3.0;
- Mesh generating using snappyHexMesh in to OpenFOAM;
- Set boundary/initial conditions (BC/IC);
- Solver : **interFoam** .

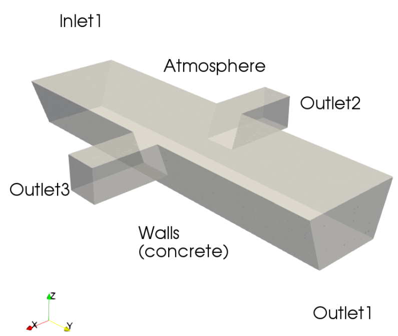

Figure 1: