

# Numerical simulation of flow characteristics over spillway 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*

February 17, 2020

## Abstract

This case study demonstrates the simulation of flow characteristics over spillway. In this case study, the flow field over the spillway simulated using open-source code (OpenFOAM). A spillway is a structure and the application of spillways is used to control flows from the dam. Spillway ensures that the water does not overflow and destroy or damage the dam. The numerical simulations for tracking the free surface are carried out using a volume of fluid (VOF) method. The simulations are performed using OpenFOAM-v7. The simulation results for velocity, water distribution, and pressure are analyzed along the spillway surface which are obtained from the simulation.

## Problem statement

Solving incompressible, transient flow in a 2D domain as shown in Figure 1. Water enters with flow rate ( $Q$ ,  $\text{m}^3/\text{s}$ ). The geometrical parameters are shown as in the Figure. In this case, two phases flow simulation approaches are considered.

- Creating background a 2D mesh by using blockMesh utility;
- Creating a surface file (stl) using SALOME-9.3.0 (spillway.stl);
- Mesh generating using snappyHexMesh in to OpenFOAM;
- Set boundary/initial conditions (BC/IC);
- Solver : **interFoam** .

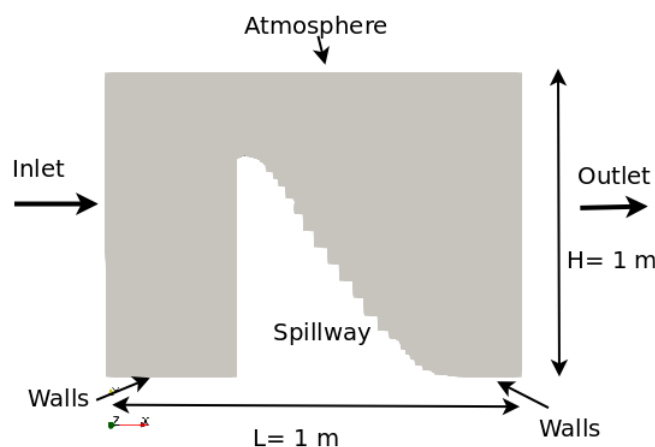


Figure 1: