

# Numerical simulations of flow characteristics over a triangular hump 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 14, 2020

## Abstract

This case study demonstrates the simulation of flow over a triangular hump. This is an example of hydraulic problems such as flow over a triangular hump (weir) and this kind of problems have various engineering applications such as dams during flood times and discharging excess water from rivers. In the present study, the flow is considered incompressible, transient and isothermal over a triangular hump in a channel. Water enters into triangular hump also referred to as gravity flow or free-surface flow. Numerical simulations are carried out to investigate the flow characteristics over a triangular hump in the channel flow. The free surface of the water is tracking using Volume of Fluid (VOF) method approach. The geometry and meshing of a triangular hump are created using SALOME-9.3.0, blockMesh and snappyHexMesh tools. The simulations are performed using OpenFOAM-v6. The simulation results of velocity and pressure profiles, and volume fraction are analyzed flow over a triangular hump.

## Problem statement

Numerical simulations of hydraulic problems such as flow over a triangular hump in a channel are performed using computational fluid dynamics. Solving incompressible, transient and isothermal flow over a triangular hump 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 0.002 m of height channel) and then water enters in to the channel. The geometrical parameters are shown as channel (1000 cm x 30 cm x 60 cm) and hump (50 cm x 30 cm x 7 cm).

- 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** .

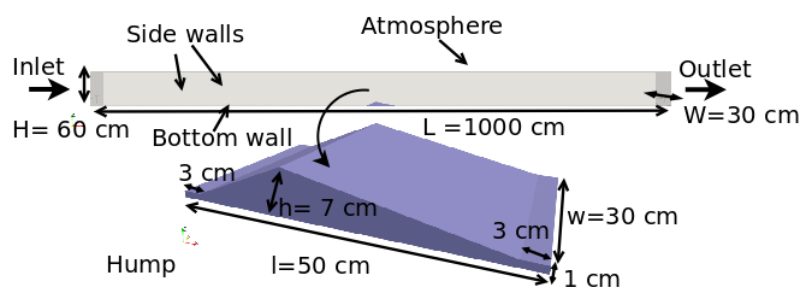


Figure 1: