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 king 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 toots. 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.

Keywords: Hump, channel flow, Two-phase flow, Volume of Fluid, CFD, OpenFOAM

1 Introduction

The purpose of this case study is to learn in OpenFOAM software [1] to the new users and to understand three dimensional flow over a triangular hump in a channel. Numerical simulations of hydraulic problems such as flow over a triangular hump in a channel are performed using computational fluid dynamics [2, 3, 4]. In the simulations, a pressure-based finite volume method is used for incompressible, isothermal and transient flow. The simulations are performed using OpenFOAM-v6 [1]. In this case, two phases flow simulation approaches are considered with using **interFoam** solver in OpenFOAMO-v6 [1]. This case study demonstrates how to do the following:

- Set up a problem case;
- Creating a surface file (stl) using SALOME-9.3.0 (plate.stl, domain.stl);
- Created the geometry import in OpenFOAM;
- Creating background a 3D mesh by using blockMesh utility;
- Mesh generating using snappyHexMesh in to OpenFOAM;
- solve a transient problem using the VOF model;
- Set up the properties of the fluids;
- Initialize the flow;
- Consider the turbulence model for turbulent flow regime;
- Post processing the case for results.

2 Problem statement

This case considers the flow over a triangular hump in a channel. Water enters into channel with flow rate (Q, m^3/s). In this case, two phases of flow simulation approaches are considered for taking the free surface using VOF. Figure 1 shows the geometry of the channel with triangular hump considered in the present study. OpenFOAM solves three-dimensional (3D) geometry in this case. 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 into the channel. The geometrical parameters and flow conditions are shown as in Table 1 and Table 2 respectively.

Table 1: Details of geometrical parameters

Parameters	Value
Height the channel (H), cm	60
Length of the channel (L), cm	1000
Width of the channel(W), cm	30
Height the Hump (h), cm	7
Length of the Hump (l), cm	50
Width of the Hump (w), cm	30

Table 2: Details of fluids property

Parameters	Value
Dynamic viscosity, water (μ_1), Pa.s	1e-03
Dynamic viscosity, air (μ_2), Pa.s	1.48e-05
Density, water (ρ_1), kg/m^3	1000
Density, air (ρ_2), kg/m^3	1
Surface tension, water-air (σ), N.m	0.07

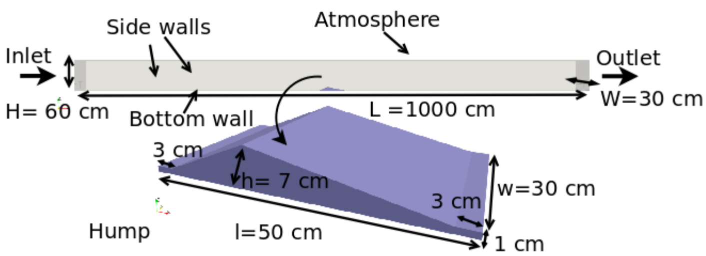


Figure 1: Schematic diagram of geometry

3 Mathematical modeling

InterFoam (solver) solves the Navier-Stokes equations for isothermal and incompressible flow. It means that all material properties are constant in the region filled by one of the two fluid except at the interphase.

3.1 Continuity equation

The continuity equation (constant-density) is defined as:

$$\frac{\partial u_j}{\partial x_j} = 0 \tag{1}$$

3.2 Momentum equation

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\tau_{ij} + \tau_{t_{ij}}\right) + \rho g_i + f_{\sigma i} \tag{2}$$

where, u represents the velocity, ρ density of the fluid, p the pressure, g_i the gravity and $\tau_{t_{ij}}$, τ_{ij} and $f_{\sigma i}$ are the turbulent, viscose stresses and the surface tension respectively.

The density of mixture fluid (ρ) is defined as (Equation- 3):

$$\rho = \alpha\rho_1 + (1-\alpha)\rho_2 \tag{3}$$

The value of α is 1, it means that the fluid 1 occupied with the density ρ_1 and the fluid 2 is occupied with the density ρ_2 with 0 the value of α. The α varies between 0 and 1 at the interphase between the two fluids.

The volume fraction α is specified as (Equation- 4):

$$\alpha = \begin{cases} 1 & \text{water phase} \\ 0 < \alpha < 1 & \text{air - water interface} \\ 0 & \text{air phase} \end{cases} \tag{4}$$

The Continuum Surface Force (CSF) approach is used to modelled the surface tension ($f_{\sigma i}$) [5, 6]. It can be calculated using Equation- 5:

$$f_{\sigma i} = \sigma\kappa\frac{\partial\alpha}{\partial x_i} \tag{5}$$

σ and κ are the surface tension constant and radius of the curvature respectively. The curvature can be estimated using Equation- 6 [5, 6].

$$\kappa = -\frac{\partial n_i}{\partial x_i} = -\frac{\partial}{\partial x_i}\left(\frac{\partial\alpha/\partial x_i}{|\partial\alpha/\partial x_i|}\right) \tag{6}$$

3.3 Volume of fluid

In order to investigate where the interphase between the two fluids (water, air) is, an additional equation for α is to be solved.

$$\frac{\partial\alpha}{\partial t} + \frac{\partial(\alpha u_j)}{\partial x_j} = 0 \tag{7}$$

3.4 k-ω turbulence model

k- ω turbulence model is obtained by substituting the eddy viscosity, ν_t = k/ω in the RANS equation [7].

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = \rho\tau_{ij}\frac{\partial u_i}{\partial x_j} - \beta^*\rho\omega k + \frac{\partial}{\partial x_j}\left[\left(\mu + \sigma_k\frac{\rho k}{\omega}\right)\frac{\partial k}{\partial x_j}\right] \tag{8}$$

$$\frac{\partial(\rho\omega)}{\partial t} + \frac{\partial(\rho u_j\omega)}{\partial x_j} = \frac{\alpha\omega}{k}\tau_{ij}\frac{\partial u_i}{\partial x_j} - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}\left[\left(\mu + \sigma_\omega\frac{\rho k}{\omega}\right)\frac{\partial\omega}{\partial x_j}\right] + \frac{\rho\sigma_d}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j} \tag{9}$$

3.5 Boundary conditions

Details of boundary name and corresponding boundary conditions are presented in Table 3.

Table 3: Boundary conditions

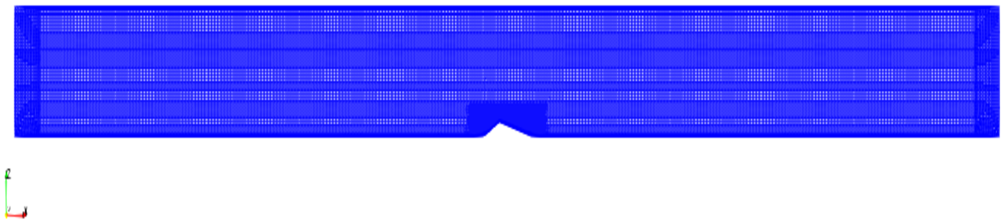
Boundary Name	Boundary condition
maxY (Wall)	No slip
minX (Inlet)	Velocity
maxX (Outlet)	Pressure
minY (Wall)	No slip
minZ (Wall)	No slip
maxZ (free surface)	Atmosphere
wall_front_face	No slip
wall_back_face	No slip

4 Simulation procedure

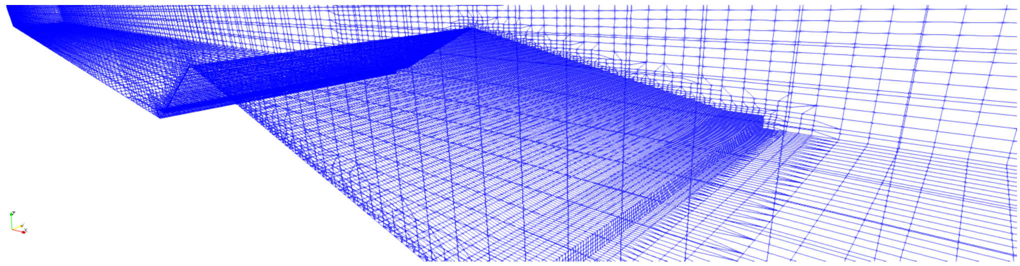
This case deals with three-dimensional turbulence simulation of water-jet. First step in setting up of an OpenFOAM case is to copy to present working directory. We need to set all require input parameters before starting the simulation. Mesh generation and implementation of boundary conditions are adopted from a base damBreak tutorial. This case study is considered with transient and turbulent flow. The solver settings are imported from damBreak tutorial (OpenFOAM/(username)-6/run/tutorials/multiphase/interFoam/ turbulence/damBreak/) from **interFoam**.

4.1 Creating geometry and mesh

- Geometry for the present problem is considered 3D dimensional domain. The geometry and mesh are generated by using the Salome, the files (.stl) is available in 'case/cad' folder in user directory (∼/case/cad).
- Mesh can be generate using 'blockMesh' and 'snappyHexMesh' utilities in to OpenFOAM.
- Figure 2 shows the isometric view of the generated mesh using 'snappyHexMesh' utility.
- All modifications for mesh with proper boundary condition with defining inlet and outlet is to be done as velocity patch and its neighbor patch (∼/case/constant/polyMesh/boundary).



(a) Mesh



(b) Zoom view

Figure 2: Computational geometry

4.2 Setting boundary conditions (BC)/Initial conditions (IC)

Files present in ‘0’ folder (∼/case/0) has been kept ’alpha.water’, ’alpha.water.org’, ’p_rgh’, ’k’,’omega’,’nut’ and ’U’ files for transient flow. Boundaries are assigned and added eight boundaries of present case in six the files, i.e. ’alpha.water’, ’p_rgh’,’k’,’omega’,’nut’ and ’U’ respectively. Details of the boundary conditions are listed in Table 4.

Table 4: Details of boundary conditions

Boundary	k	p_rgh	U	alpha.water	nut	omega
maxY	kqRWallFunction	fixedFluxPressure	fixedValue	zeroGradient	nutkWallFunction	omegaWallFunction
minX	fixedValue	fixedFluxPressure	flowRateInletVelocity	fixedValue	calculated	fixedValue
maxX	inletOutlet	zeroGradient	inletOutlet	zeroGradient	calculated	inletOutlet
minY	kqRWallFunction	fixedFluxPressure	fixedValue	zeroGradient	nutkWallFunction	omegaWallFunction
minZ	kqRWallFunction	fixedFluxPressure	fixedValue	zeroGradient	nutkWallFunction	omegaWallFunction
maxZ	inletOutlet	ptotalPressure	pressureInletOutletVelocity	inletOutlet	calculated	inletOutlet
wall_front_face	kqRWallFunction	fixedFluxPressure	fixedValue	zeroGradient	nutkWallFunction	omegaWallFunction
wall_back_face	kqRWallFunction	fixedFluxPressure	fixedValue	zeroGradient	nutkWallFunction	omegaWallFunction

4.3 Solver details

The volume of fluid (VOF) method for phase fraction is used for incompressible, isothermal, immiscible fluids based on interface capturing approach with optional mesh and geometrical reconstruction technique is used (piecewise linear interface calculation) in simulation.

In the present study, unsteady state and turbulent flow are considered. Turbulent flow model can be applied in the OpenFOAM by using 'simulationType' option in the 'turbulence Properties' file in constant folder. The 'simulationType' option can be used as '**RAS**'. In order to run unsteady state simulations, blockMeshDict, changeDictionaryDict, controlDict, decomposeParDict (for parallel computation), fvSchemes,fvSolution, snappyHexMeshDict, surfaceFeatureExtractDict and setFieldsDict files are kept in the system directory folder. '**./run**' command executes in terminal to run computations.

4.4 Post-processing

The paraFoam, it can be used to visualize the simulations results in OpenFOAM. This can be run by typing the following command line in the terminal **paraFoam** to open the ParaView software and upload the case.

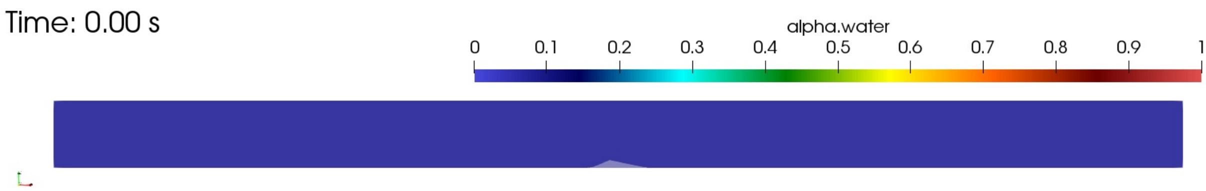
5 Results and discussion

Simulation are performed using OpenFOAM to investigate the flow over a triangular hump in a tank. Results are plotted with the help of 'paraview' ($\sim$/case/paraFoam). Simulation results are analyzed with the help of paraFoam software. Figures 3- 5 show the field α (volume fraction) for water, fields p pressure and and v velocity distribution which are obtained from simulations.

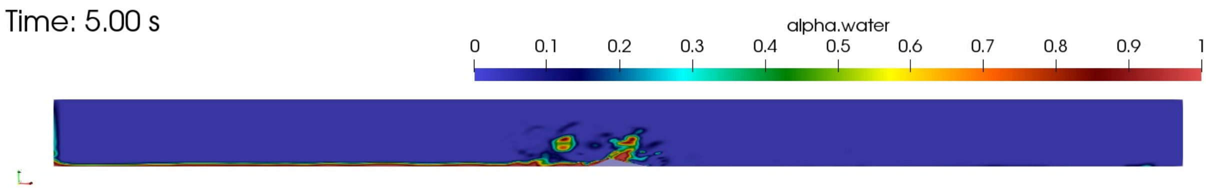
Figure 3a shows a plane in isomeric view of the domain colored by the volume fraction (red color correspond to α=1, water and blue color corresponds to α= 0, air) at inlet flow rate of 30 m^3/hr. It can be seen from the Figure that initially the water enters in the inlet channel while the tank is filled with water (0.002 m out of 0.6 m, tank height) and air.

Figures 3b to h show the field α after 5 s from the start of the water flowing over the hump. The results shows that the water flow over the hump in the channel.

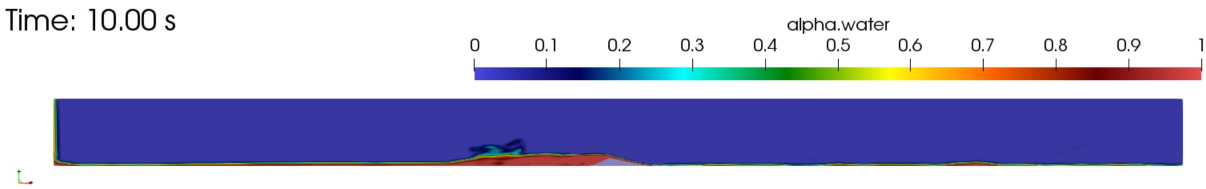
Figures 4 & 5 show the pressure and velocity distribution over the hump in the channel for various time.



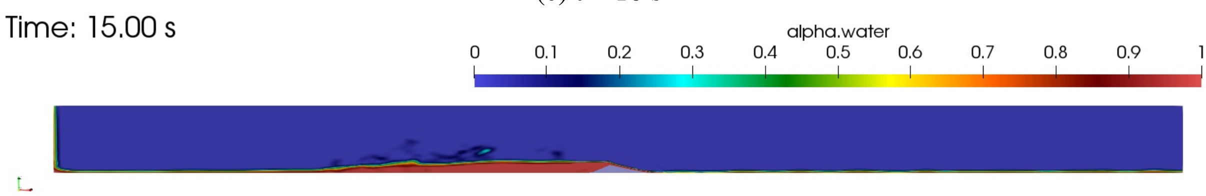
(a) t = 0 s



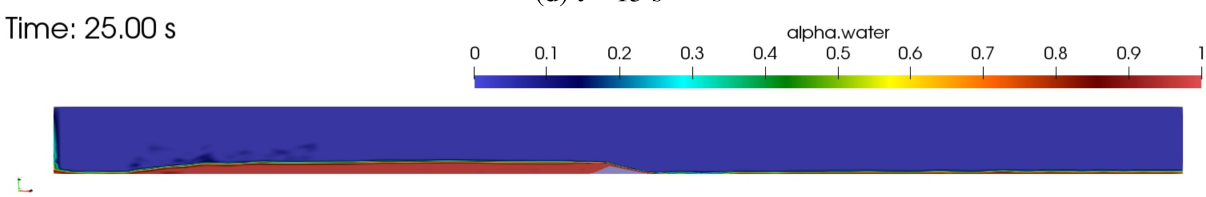
(b) t = 5 s



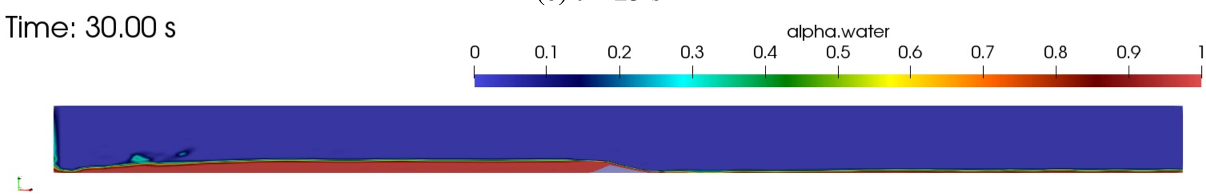
(c) t = 10 s



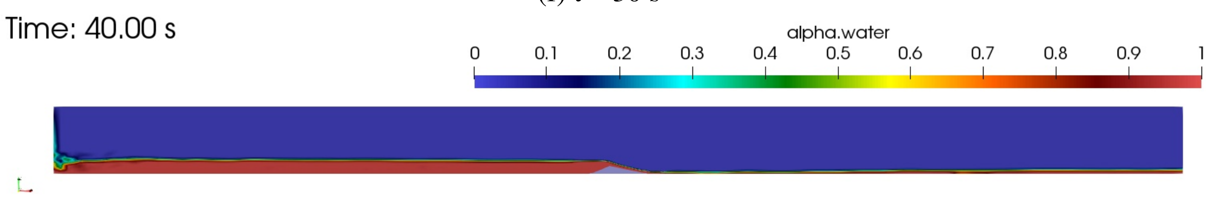
(d) t = 15 s



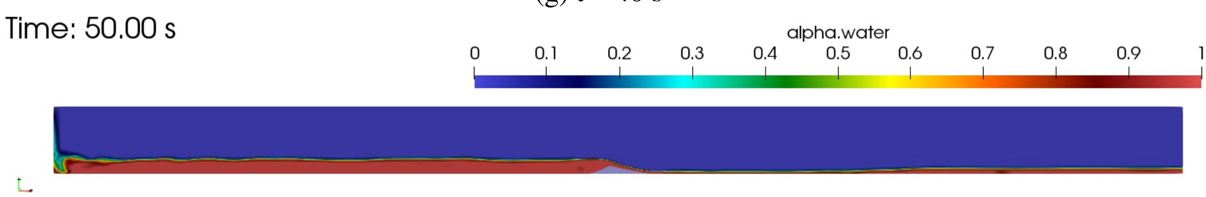
(e) t = 25 s



(f) t = 30 s

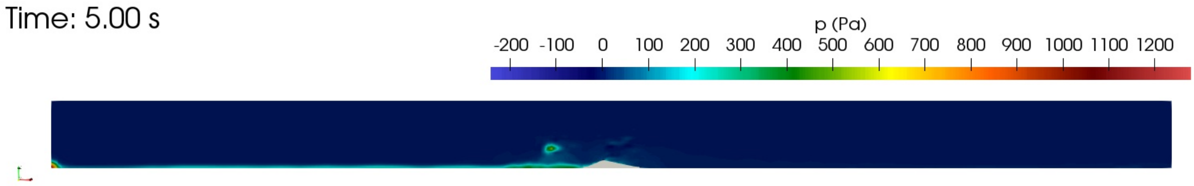


(g) t = 40 s

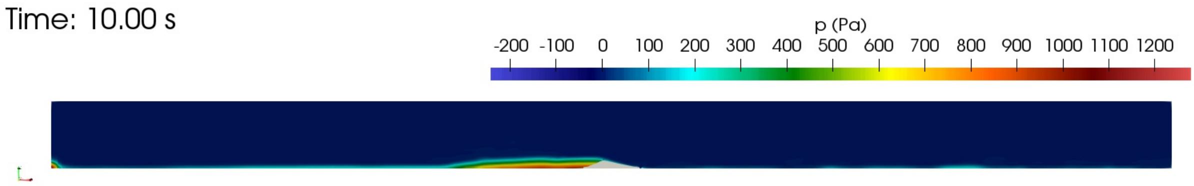


(h) t = 50 s

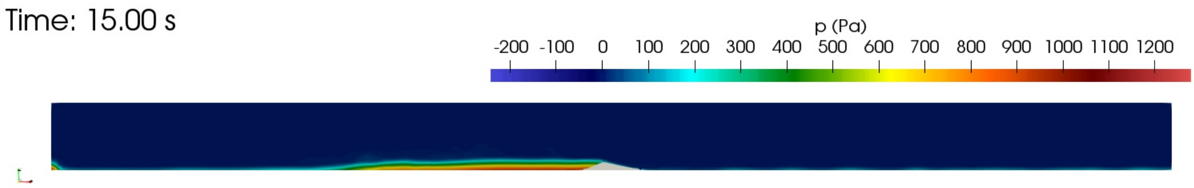
Figure 3: Field α or volume fraction of water at different time with inlet flow rate of water, **Q= 30 m^3/s** (red color correspond to α=1, water and blue color corresponds to α= 0, air).



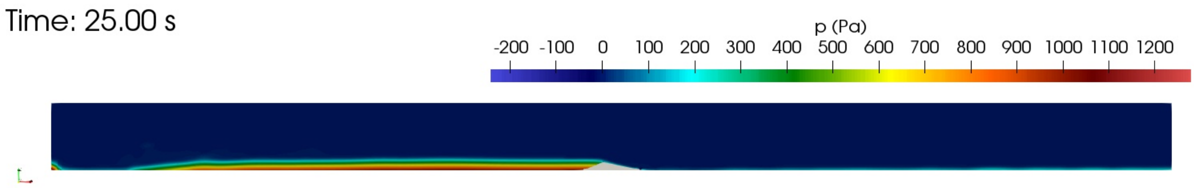
(a) t = 5 s



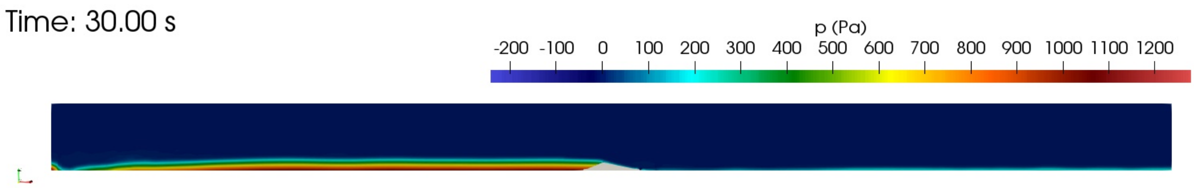
(b) t = 10 s



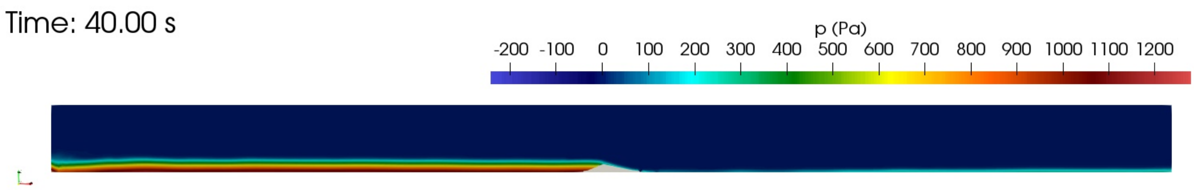
(c) t = 15 s



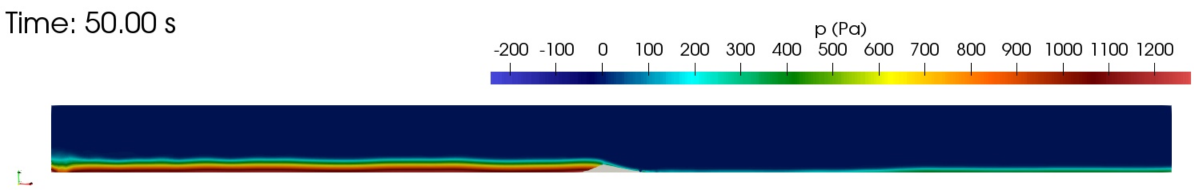
(d) t = 25 s



(e) t = 30 s

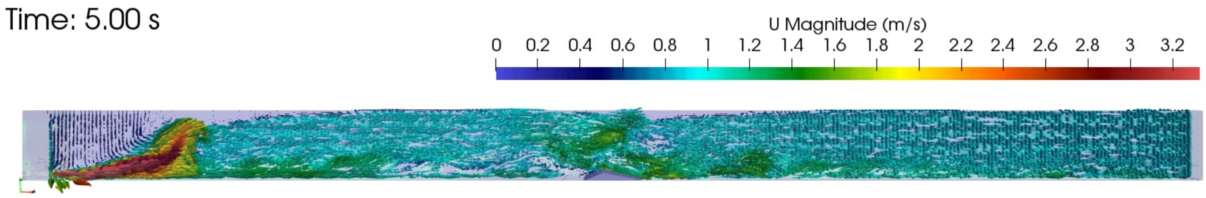


(f) t = 40 s

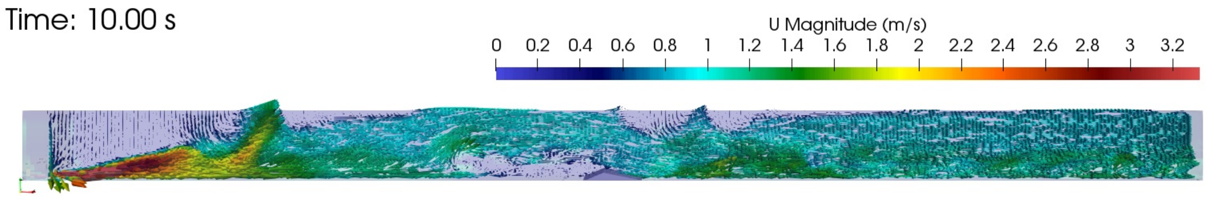


(g) t = 50 s

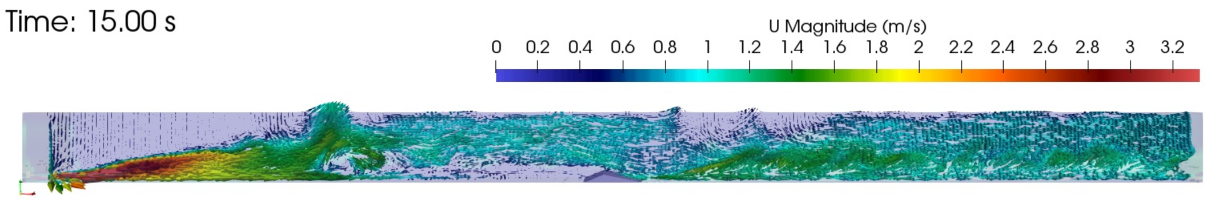
Figure 4: Field p or presure distribution at different time with inlet flow rate of water, **Q= 30 m^3/s**.



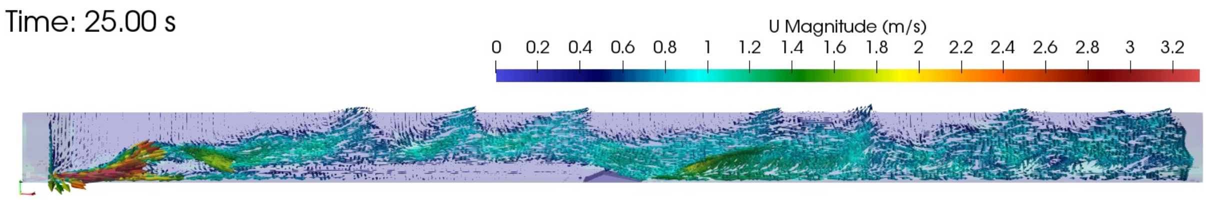
(a) t = 5 s



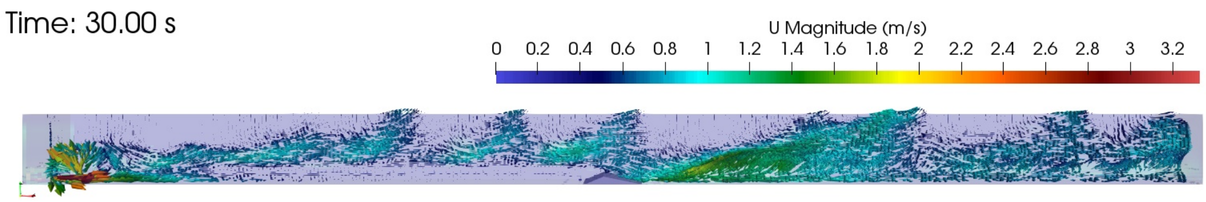
(b) t = 10 s



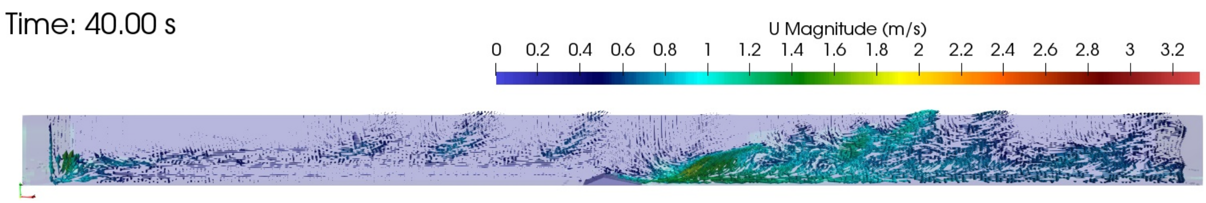
(c) t = 15 s



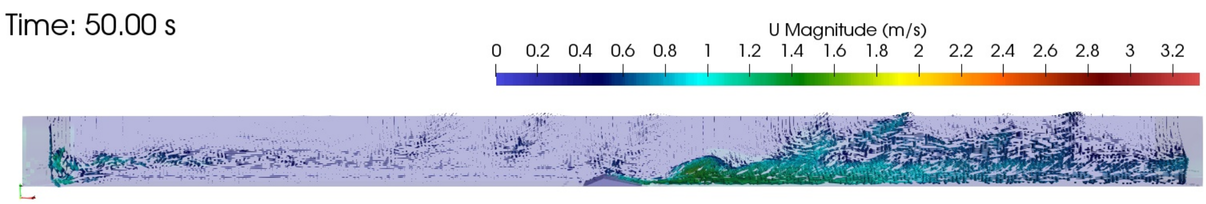
(d) t = 25 s



(e) t = 30 s



(f) t = 40 s



(g) t = 50 s

Figure 5: Field v or velocity distribution at different time with inlet flow rate of water, **Q= 30 m^3/s**.

References

[1] C. J. Greenshields, *OpenFOAM: The OpenFOAM Foundation. User Guide Version 6*. CFD Direct Limited, July. 2018.

[2] A. Ferrari, "Sph simulation of free surface flow over a sharp-crested weir," *Advances in Water Resources*, vol. 33, no. 3, pp. 270 – 276, 2010.

[3] S. Dehdar-behbahani and A. Parsaie, "Numerical modeling of flow pattern in dam spillway's guide wall. case study: Balaroud dam, iran," *Alexandria Engineering Journal*, vol. 55, no. 1, pp. 467 – 473, 2016.

[4] E. H. H. Al-Qadami, A. S. Abdurrasheed, Z. Mustaffa, K. W. Yusof, M. Malek, and A. A. Ghani, "Numerical modelling of flow characteristics over sharp crested triangular hump," *Results in Engineering*, vol. 4, p. 100052, 2019.

[5] J. Brackbill, D. Kothe, and C. Zemach, "A continuum method for modeling surface tension," *Journal of Computational Physics*, vol. 100, no. 2, pp. 335 – 354, 1992.

[6] J. A. Heyns and O. F. Oxtoby, "Modelling surface tension dominated multiphase flows using the vof approach," 2014.

[7] D. C. Wilcox, "Formulation of the k-w turbulence model revisited," *AIAA Journal*, vol. 46, no. 11, pp. 2823–2838, 2008.