FOSSEE, IIT Bombay
OpenFOAM Case Study Project
April 2025

CFD Simulation of Hydraulic Jump using OpenFOAM v2412

Karri Vijay Reddy[1], Ranjit Desai[2], Himani Garg[3],
Sameer Jadhav[4], Parees Palkar[5]

[1]Institute of Aeronautical Engineering (IARE), Hyderabad
[2]WaterUnit Infrastructure Consultancy (Guide)
[3]Lund University, Sweden (Co-guide)
[4]IIT Bombay (Co-guide)
[5]FOSSEE, IIT Bombay (Mentor)

Abstract

A two-dimensional Computational Fluid Dynamics (CFD) simulation of a hydraulic jump in a rectangular open channel was carried out using OpenFOAM v2412. The Volume-of-Fluid (VOF) solver `interFoam` was employed with the `kOmegaSST` Reynolds-Averaged Simulation turbulence model. The inflow Froude number was set to $Fr_1 = 6.0$, producing a steady hydraulic jump with a theoretically predicted conjugate depth of $y_2 = 0.80$ m. A sharp-crested weir was incorporated at $x = 15.0$ m downstream as a backwater control structure. The weir height of $h = 0.386$ m was determined through an iterative Excel-based calculator using the Rehbock weir discharge formula, validated against a standalone weir-only CFD simulation. The hydraulic jump simulation confirmed jump formation, a recirculation roller zone, and submerged jump conditions consistent with analytical predictions. Energy dissipation efficiency was analytically computed at 56.4%.

Keywords: hydraulic jump, OpenFOAM, interFoam, Volume-of-Fluid (VOF), kOmegaSST, free surface flow, open-channel hydraulics, sharp-crested weir, Rehbock formula, Froude number, energy dissipation, computational fluid dynamics, FOSSEE.