

Numerical simulations of flow over a flat plate in OpenFOAM and Fluent

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

December 17, 2019

Abstract

This case study demonstrates the flow over a flat plate. 2D case model (geometry and mesh) is made with blockMesh meshing tool/utility. The flow is considered steady-state, isothermal and laminar. The simulations are performed using OpenFOAM-v6 and Ansys Fluent-v19.2. The hydrodynamics of flow over a flat plate is investigated. The simulations results of OpenFOAM and Ansys-Fluent are compared and analyzed.

Problem Statement

Solving incompressible flow in a 2D (planer) flat plate for steady state (Figure 1).

- Creating a 2D planer geometry and mesh by using blockMeshDict
- Set boundary/initial conditions (BC/IC) : Velocity 0.01 - 0.10 m/s
- Solver : **simpleFoam**

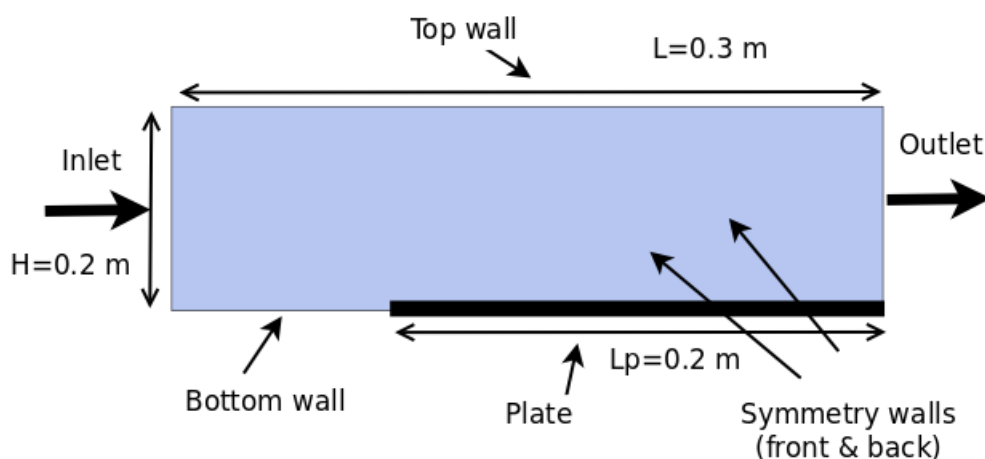


Figure 1: