Numerical simulations of fluid flow and heat transfer in a heater 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 1, 2020

Abstract

This case study demonstrates the simulation of a heater. It is an example of heat transfer between two parallel plates (heater). The present case also describes the conjugate heat transfer in multi-region (solid and water) and solver (chtMultiRegionFoam) in open source CFD package OpenFOAM. 3D case model (geometry and mesh) is made with blockMesh meshing tool/utility. The flow is considered transient, non-isothermal and laminar. The simulations are performed using OpenFOAM-5.x. The hydrodynamics of flow between two parallel plates is investigated. The temperature profile is analyzed obtained from the simulation.

Problem Statement

The geometric parameters of the domain such as height, width and depth are considered with 60x40x40 (units, 0.40 m x 0.16 m x 0.20 m) respectively. The solid and water region are defined with topoSet in a 3D environment. The fluid is considered water. Initially, temprature is patched with value of 300 K.

- Creating a 3D mesh by using blockMesh utility;

- Set physical properties (transportPropeties);

- Set boundary/initial conditions (BC/IC);

- Set numerical schemes, solver parameters and control parameters;
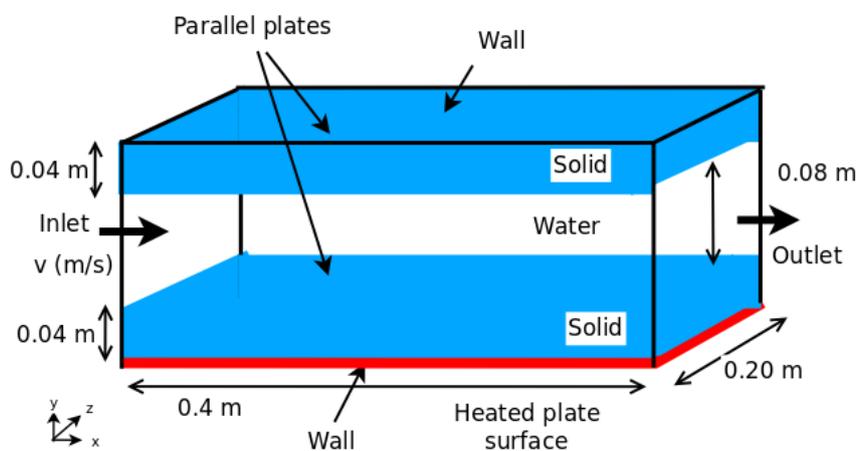
- Solver- **chtMultiRegionFoam** .

Figure 1: