Numerical simulation of heating pipe 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

December 9, 2019

Abstract

This case study demonstrates heat flow through a pipe. 2D case model is made with blockMesh meshing tool. The present case also describes the implementation of convective heat flux boundary condition and solver in open source CFD package OpenFOAM. The study is carried out using OpenFOAM-5x. The boundary conditions are used in numerical investigations of temperature. Steady, laminar and fully developed flow in a pipe is considered and has been solved by using 'simpleThermFoam' solver. The temperature profile is analyzed obtained from the simulation.

Problem statement

Solving incompressible flow in a 2D axisymmetrical pipe in the present study as shown in Figure 1. This case explains setting up OpenFOAM case in the OpenFOAM repository.

Solver creation

- To create the temperature fields and to read the physical properties: createFields.H

- Modify solver to add temperature equation: simpleThermFoam.C

- Create boundary condition: convectiveHeatFluxFvPatchScalarField.C and convectiveHeatFluxFvPatchScalarField.H

Case set-up

- Creating a 2D mesh by using blockMesh

- Set boundary/initial conditions (BC/IC)

- Set physical properties (transportPropeties)

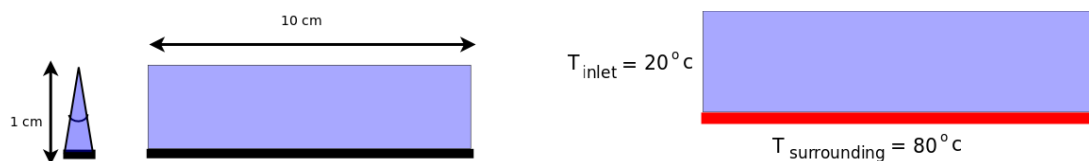- Set numerical schemes, solver parameters and control parameters

- Solver-simpleThermFoam

Figure 1: