

# Numerical simulations of double pipe heat exchanger with parallel flow in 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*

February 16, 2020

## Abstract

This case study demonstrates the simulation of a double pipe heat exchanger with the parallel flow in or parallel flow. This type of heat exchanger (double pipes) is most commonly used in refrigeration, space heating, air conditioning, sewage treatment, power plants, petrochemical plants, petroleum refineries, chemical plants, and natural gas processing and it is one of the simplest types of heat exchangers. The present case also describes the conjugate heat transfer in multi-region (solid and fluids). The flow is considered steady-state/unsteady state, non-isothermal and laminar/turbulent. The simulations are performed using OpenFOAM-v5. The hydrodynamics of flow between both the pipes is investigated. The temperature and velocity profile is analyzed for parallel or co-counter flow in the pipes which are obtained from the simulation.

## Problem Statement

The geometric parameters of the domain such as length and diameter of pipes are considered with 2 m and the diameters ( $R_i/R_o$ ) of inside and outside pipe are 6 mm and 20 mm respectively. The solid and two fluids region are defined in a 3D environment. Initially, the fluids temperature are considered 60 (hot/inside) and 15 ° C (cold/outside).

- Creating a 3D mesh by using blockMesh utility;
- Set physical properties (transportProperties);
- Set boundary/initial conditions (BC/IC);
- Set numerical schemes, solver parameters and control parameters;
- Solver- **ChtMultiRegionSimpleFoam** .

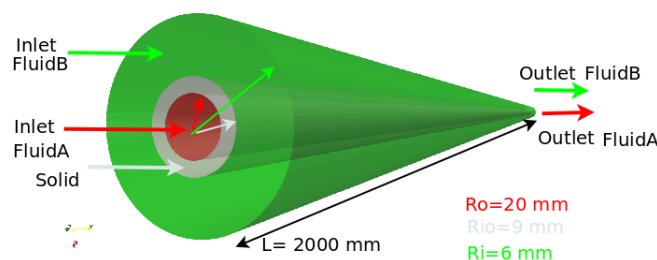


Figure 1: