

CFD analysis of fluid flow and heat transfer in a shell and tube heat exchange 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

January 19, 2020

Abstract

This case study demonstrates the simulation of a shell and tube heat exchanger. This type of heat exchanger (shell and tube) is most commonly used in oil refineries and other large chemical processes and where it suits for higher-pressure applications. It is an example of heat transfer between two fluids. The present case also describes the conjugate heat transfer in multi-region (solid and fluids). The flow is considered steady state, non-isothermal and turbulent. The simulations are performed using OpenFOAM-v7. The hydrodynamics of flow between tube and shell is investigated. The temperature and velocity profile is analyzed obtained from the simulation.

Problem Statement

The geometric parameters of the domain such as length and diameter of shell are considered with 0.250 m and 0.08 m and the diameters (in/out) of tube are 0.012 and 0.020 m respectively. The solid and two fluids region are defined in a 3D environment. Initially, the fluids temperature are considered 700 (600, 500) and 300 K.

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

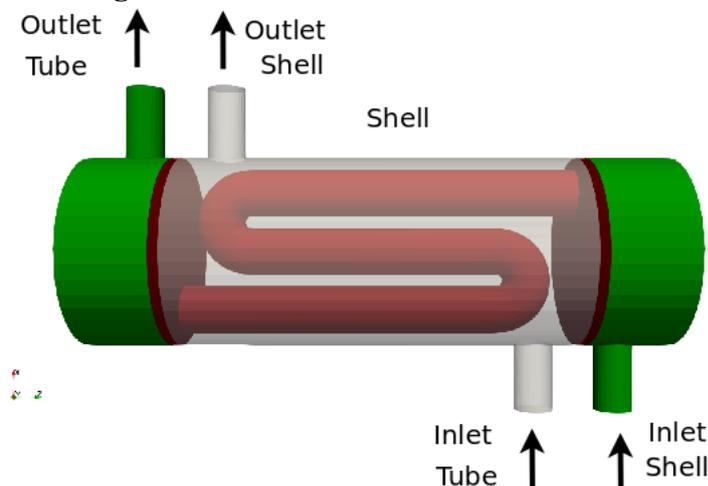


Figure 1: