# Numerical simulations of tubular reactor hydrodynamics 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 14, 2019

## Abstract

This case study demonstrates to learn how to solve specific equations. The aim of the present study is to simulate one-dimensional flow in a tubular reactor without considering chemical reaction on porous media in steady state. 1D case model is made with blockMesh meshing tool. This study is performed using OpenFOAM-5x. It's a purpose to describe dealing only hydrodynamics of tubular reactor without considering chemical reaction and implementation of new solver (tubeFoam) with open-source CFD package OpenFOAM. The simulation results are obtained and have been analyzed.

## Problem Statement

The hydrodynamics of tubular reactor with 1D heterogeneous catalysis is on porous media without chemical reaction for steady-state.

- Creating a 1D symmetrical mesh by using blockMeshDict;

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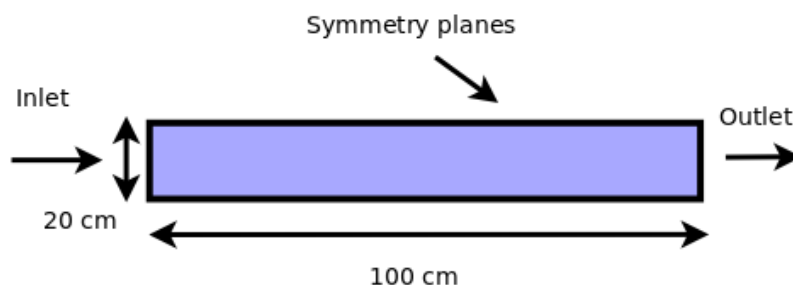
- Set Physical properties;

- Brief explanation of solver creation;

- Solver-**tubeFoam** .



Figure 1: Tubular reactor