

# Numerical simulation of methane combustion with oxygen in air 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*

February 18, 2020

## Abstract

This case study demonstrates the simulation of combustion of methane and air in the mixing chamber (tubular reactor). The combustion takes place when fuel reacts with the oxygen in the air to produce heat. Complete combustion occurs when none of the carbon and hydrogen compounds left unburned. The aim of the present study is to understanding multi species and reaction modeling in a tubular reactor with a chemical reaction in an unsteady and transient flow. The present case also describes how to model combustion phenomena in which methane reacts with oxygen involves. The simulations are performed using OpenFOAM-v5. The concentration profiles of the reactants and products are analyzed in the reactor which are obtained from the simulation.

## Problem Statement

The geometric parameters of the combustion chamber domain (3D tubular reactor) such as length and diameter are considered 2000 mm with 250 mm respectively. The fuel inlet diameter ( $D_i$ ) and air inlet diameter ( $D_o - D_i$ ) are 10 mm and 240 mm respectively. The fluid region is defined in a 3D environment. Initially, the fuel and air ignition temperature is considered 1900 K or 1627 °C (hemisphere of 4.5 mm diameter at the inlet section).

- 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- **reactingFoam** .

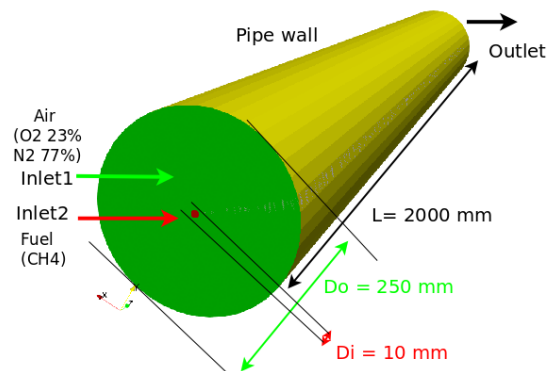


Figure 1: