FOSSEE, IIT Bombay
Case Study Project
July 30, 2024

Numerical simulation of methane jet using OpenFOAM

Aditya TR

Department of Aerospace Engineering, Amrita Vishwa Vidyapeetam,

Coimbatore, Tamilnadu, India-641112

Mentor: Nishit Pachpande

FOSSEE, IIT Bombay

Guide: Dr. Harikrishnan S

Division of Mechanical Engineering, School of Engineering,

Cochin Univerisity of Science and Technology, Kochi, Kerala, India-682022

Abstract

The objective of the present study is to numerically simulate methane jets in open source CFD package OpenFOAM. This report will help reader to understand the step-by-step procedure involved in simulating methane jets and also explain the details regarding initial and boundary conditions, solver settings etc. Geometry and mesh has been generated using 'blockMesh' utility available in OpenFoam and flow has been simulated using 'reactingFoam' solver. Two different conditions have been considered here viz. subsonic and supersonic jets. Numerical method has been validated against the benchmark experiments by Birch et al. (1984).

References

Birch, A., Brown, D., Dodson, M. & Swaffield, F. (1984), 'The structure and concentration decay of high pressure jets of natural gas', *Combustion Science and technology* **36**(5-6), 249–261.