Flow Past a Square Cylinder Using OpenFOAM

Ananthya Sankar
Department of Mechanical Engineering
Vellore Institute of Technology (VIT)
Vellore, Tamil Nadu, India
Software: OpenFOAM v13

ABSTRACT

The objective of the present project is to investigate incompressible laminar flow past a square cylinder using the open-source CFD package OpenFOAM. Flow past bluff bodies is a classical benchmark problem in fluid mechanics due to the formation of an unsteady wake and periodic vortex shedding behind the body. The study of such flows is important in many engineering applications including civil structures, heat exchangers, automotive systems and aerospace components.

A two-dimensional computational domain containing a square cylinder will be considered at Reynolds number 100. The geometry and mesh will be generated using the blockMesh utility available in OpenFOAM. The flow field will be solved using the foamRun solver with the incompressibleFluid module available in OpenFOAM v13. The study will focus on analysing velocity distribution, pressure distribution, vorticity contours, streamline patterns, drag coefficient, lift coefficient and vortex shedding frequency.

The numerical results will be compared with benchmark data available in the literature to validate the simulation. The project aims to provide a detailed understanding of wake dynamics behind bluff bodies and demonstrate the use of OpenFOAM for external aerodynamic flow simulations. The primary objective of this study is to capture the vortex shedding behaviour of flow past a square cylinder and validate the computed aerodynamic coefficients against published benchmark data.

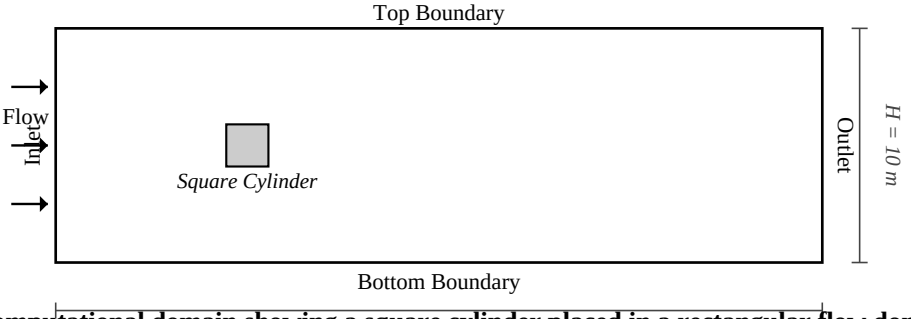

Figure 1: Computational domain showing a square cylinder placed in a rectangular flow domain with inlet, outlet and wall boundaries clearly labelled.

Table 1: Details of Geometry and Flow Conditions

Parameter	Value
Cylinder Side Length	1 m
Domain Length	20 m
Domain Height	10 m
Inlet Velocity	0.1 m/s
Kinematic Viscosity	0.001 m²/s
Reynolds Number	100