FOSSEE, IIT Bombay
OpenFOAM Case Study Project
December 2025

FORCED CONVECTION HEAT TRANSFER OVER A HEATED CYLINDER IN OpenFOAM

Kartik Bajwan[1], Bibek Dhungana[2], Chandan Bose[3], Manjil Sitoula[4]

[1] B.Tech, Department of Chemical Engineering, Indian Institute of Technology Ropar, India

[2] Department of Aerospace Engineering, Tribhuvan University, Nepal

[3] Department of Aerospace Engineering, University of Birmingham, United Kingdom

[4] FOSSEE, Indian Institute of Technology Bombay, India

ABSTRACT

Laminar flow over a heated circular cylinder has been widely studied because of its importance in understanding flow instability and its relevance to many engineering applications such as hot-wire anemometry heat exchangers, electronic cooling. In this work, forced convection heat transfer over a heated circular cylinder is numerically investigated using OpenFOAM. Simulations are run for Reynolds numbers Re = 50, 100, and 150 with a constant surface temperature applied on the cylinder. The incompressible Navier–Stokes and energy equations are solved on a fixed body-fitted mesh, and the resulting flow behavior is analyzed by the vortex shedding phenomena followed by velocity, vorticity , and streamlines contour plots. Drag and Lift force coefficients are also calculated using OpenFOAM function objects.

Using a Power Spectral Density (PSD) approach developed in Python, the vortex shedding frequency is calculated from the lift coefficient time history. Finally, the variation of the Strouhal number with Reynolds number is obtained and compared with published results, showing good agreement.