

FOSSEE – CFD Learning Program

Numerical Investigation Of Mixed Convective Heat Transfer Enhancement Using Rectangular Fins Using OpenFOAM

Likhith Kammara¹, Evan Fernandes², Parees Palkar²

¹Department of Chemical Engineering, Rajiv Gandhi University of Knowledge Technologies – RK Valley, A.P., India

²FOSSEE Project, Indian Institute of Technology Bombay, Mumbai, Maharashtra, India

Abstract

Heat transfer enhancement using extended surfaces is an important technique for improving the thermal performance of compact heat-transfer systems. This study investigates mixed convection heat transfer in a rectangular air channel with and without four longitudinal fins using computational fluid dynamics (CFD). The computational domain measures $0.6 \text{ m} \times 0.3 \text{ m} \times 0.1 \text{ m}$, with a heated base plate of $0.6 \text{ m} \times 0.3 \text{ m} \times 0.001 \text{ m}$. Each fin has dimensions of $0.6 \text{ m} \times 0.025 \text{ m} \times 0.001 \text{ m}$. Air is used as the working fluid, and simulations are performed at Reynolds numbers of 1000, 1500, 3000, and 6000 under constant heat-flux conditions.

The simulations are carried out using OpenFOAM 9. The computational domain is discretized using background hexahedral meshing with surface-based refinement using snappyHexMesh. A corresponding no-fin configuration is also simulated to quantify the heat-transfer enhancement produced by the fins. A mesh-independence study is conducted by comparing outlet temperature and heat-transfer characteristics for progressively refined meshes.

The temperature, velocity, and pressure fields are post-processed using ParaView. The average Nusselt number is calculated to characterize convective heat transfer, while the ratio of finned to no-fin Nusselt numbers is used to quantify enhancement. The results demonstrate the effects of Reynolds number and longitudinal fins on the thermal and flow behaviour, establishing a CFD-based methodology for evaluating heat-transfer enhancement in finned air channels.

Keywords: Computational Fluid Dynamics (CFD), OpenFOAM 9, Mixed Convection, Forced Convection, Longitudinal Fins, Heat Transfer Enhancement, Nusselt Number, Reynolds Number, Mesh Independence, Air Channel.