

# **Validating non-isothermal mechanical ventilation in a generic closure using OpenFOAM**

Agathiyan V<sup>1</sup>, Nikhil Chitnavis<sup>2</sup> and Harikrishnan S<sup>3</sup>

<sup>1</sup>Undergraduate Student, Aerospace Engineering, Vellore Institute of Technology, Sehore, Madhya Pradesh - 466 114

<sup>2</sup> Research Associate, Department of Applied Mechanics, IIT Madras, Tamil Nadu - 600036

<sup>3</sup>Assistant Professor, Division of Mechanical Engineering, School of Engineering, Cochin University of Science and Technology, Kochi, Kerala - 682022

## **ABSTRACT**

This report presents the OpenFOAM implementation and validation of a benchmark non-isothermal mechanical ventilation case reported by Kang and Van Hooff [1]. The simulations were performed using the *buoyantBoussinesqSimpleFoam* solver together with the SST  $k-\omega$  turbulence model to predict indoor airflow and temperature distribution inside a mechanically ventilated closure. Experimentally measured inlet velocity, temperature, and turbulence data were applied using prescribed boundary conditions to accurately reproduce the inlet jet behavior. The simulation results for dimensionless velocity magnitude, dimensionless air temperature, and turbulent kinetic energy (TKE) were compared with the available experimental and simulation data. This work demonstrates a reproducible OpenFOAM framework for indoor ventilation simulations and highlights the importance of accurately defining inlet boundary conditions for reliable CFD predictions.