FOSSEE CFD-OpenFOAM Semester Long Internship Spring 2026

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

Agathiyan V[1], Nikhil Chitnavis[2] and Harikrishnan S[3]

[1]Undergraduate Student, Aerospace Engineering, Vellore Institute of Technology, Sehore, Madhya Pradesh - 466 114

[2] Research Associate, Department of Applied Mechanics, IIT Madras, Tamil Nadu - 600036

[3]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 airflow inside a mechanically ventilated enclosure was simulated with the buoyantBoussinesqSimpleFoam solver coupled with the SST k–ω turbulence model. Experimentally measured inlet velocity, temperature, and turbulence data were applied using prescribed boundary conditions to accurately reproduce the inlet jet behavior. Numerical predictions of airflow velocity, temperature distribution, and turbulent kinetic energy were compared with published experimental observations and reference CFD results. 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.

1. INTRODUCTION

The prediction of indoor airflow and temperature distribution is important in the design of mechanical ventilation systems used in buildings for thermal comfort, air quality control, and energy efficiency. Indoor airflow is influenced by both forced convection from supply air jets and natural convection caused by temperature differences, resulting in complex flow patterns such as recirculation and thermal stratification.

Computational Fluid Dynamics (CFD) offers an efficient alternative by enabling detailed examination of airflow and temperature fields under different operating conditions. Among the available CFD approaches, Reynolds-Averaged Navier–Stokes (RANS) models are commonly employed for engineering applications because they offer reasonable prediction accuracy while maintaining manageable computational cost. The reliability of CFD simulations, however, depends strongly on the specification of boundary conditions and turbulence modelling strategies.

The present work reproduces a benchmark non-isothermal ventilation case using OpenFOAM. The study focuses on implementing prescribed inlet boundary conditions and validating the predicted velocity, temperature, and turbulence fields against available experimental and numerical reference data. The use of an open-source CFD platform also ensures that the simulation methodology remains transparent and reproducible for future studies.

2. PROBLEM STATEMENT

The objective of this project is to reproduce a benchmark mechanically ventilated enclosure case and evaluate the predictive capability of OpenFOAM for non-isothermal indoor airflow simulations [1, 2]. The study focuses on predicting airflow distribution, temperature variation, and turbulence behaviour inside a three-dimensional mechanically ventilated enclosure. The airflow is modelled as steady, incompressible, and turbulent, while buoyancy effects due to temperature differences are included using the Boussinesq approximation. The SST k–ω turbulence model is used to find the interaction between forced convection from the inlet jet and natural convection caused by thermal gradients inside the enclosure.

To accurately reproduce the experimental airflow behavior, experimentally measured inlet velocity, temperature, and turbulence quantities are prescribed at the inlet boundary. The numerical predictions obtained using OpenFOAM are then validated against the experimental data reported by Gresse *et al.* [2]. The schematic diagram of the computational domain considered in the present study is shown in Fig. 1.

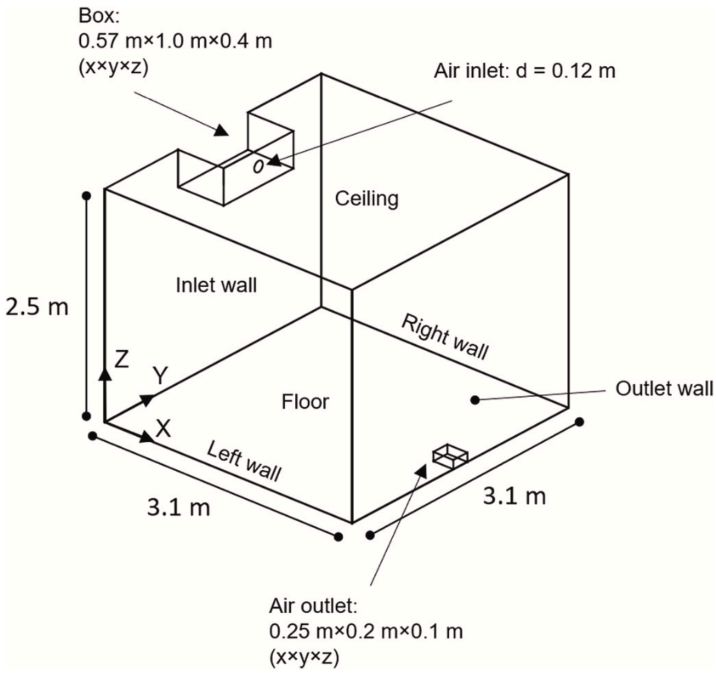


Fig. 1: Schematic of the computational domain and coordinate system.

3. NUMERICAL METHODS

3.1 Governing Equations and Boundary Conditions

To simulate non-isothermal indoor airflow, we solve the steady-state Reynolds-Averaged Navier-Stokes (RANS) equations. The coupling between air velocity and temperature variations is modeled by solving the following set of governing equations:

Continuity equation

$$\nabla \cdot \vec{u} = 0 \tag{1}$$

Momentum equation

$$\nabla \cdot (\vec{u}\,\vec{u}) = -\frac{1}{\rho}\nabla p + \nabla \cdot (\nu_{eff} \nabla \vec{u}) + g\beta(T - T_{ref}) \tag{2}$$

Energy equation

$$\nabla \cdot (\vec{u}\, T) = \nabla \cdot (\alpha_{eff} \nabla T) \tag{3}$$

Turbulent Kinetic Energy (k)

$$\nabla \cdot (\vec{u} k) = \nabla \cdot (\nu_{eff} \nabla k) + Pk - \beta^* k\omega \tag{4}$$

Specific Dissipation Rate (ω)

$$\nabla \cdot (\vec{u}\omega) = \nabla \cdot (\nu_{eff} \nabla \omega) + \alpha(\frac{\omega}{k})Pk - \beta\omega^2 \tag{5}$$

where:

- $\vec{u}$ = velocity vector (m/s)
- ρ = fluid density (kg/m^3)
- p = pressure (Pa)
- ν_{eff} = effective kinematic viscosity (m^2/s)
- g = gravitational acceleration (m/s^2)
- β = thermal expansion coefficient (1/K)
- T = local temperature (K)
- T_{ref} = reference temperature (K)
- α_{eff} = effective thermal diffusivity (m^2/s)
- k = turbulent kinetic energy (m^2/s^2)
- ω = specific dissipation rate (1/s)
- Pk = production term of turbulent kinetic energy
- α and β = turbulence model constants

The inlet boundary conditions were implemented using experimentally measured velocity, temperature, and turbulence data. These quantities were prescribed at the inlet using the *timeVaryingMappedFixedValue* boundary condition available in OpenFOAM. This approach allows spatially varying inlet profiles to be mapped directly onto the inlet patch, enabling more accurate representation of the inlet jet structure and thermal distribution inside the enclosure. Table 1 summarizes the boundary conditions applied to the reference case. The detailed implementation of the specified inlet boundary conditions and the OpenFOAM configuration can be found in the Appendix.

Table. 1: Boundary Conditions for inlet, outlet and walls.

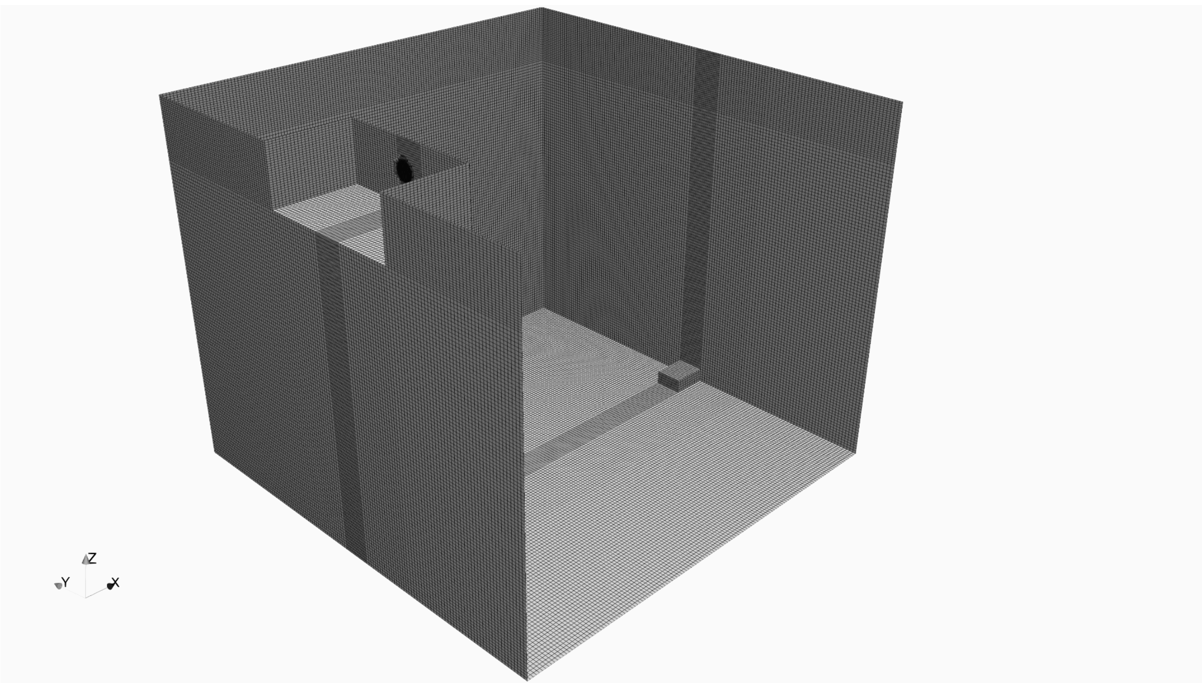
Inlet	Velocity	Prescribed Values
	Temperature	Prescribed Values
	Turbulence (k, ω)	Prescribed Values
Walls	Velocity	No-slip
	Temperature (Inlet Wall)	295.75 K (22.6°C)
	Temperature (Outlet Wall)	293.95 K (20.8°C)
	Temperature (Ceiling, Left & Right Wall)	294.15 K (21.0°C)
	Temperature (Floor)	293.85 K (20.7°C)

Outlet	Pressure (p_rgh)	0 (fixedValue)
	Velocity	Zero gradient
Turbulence (Walls)	k	kqRWallFunction
	ω	omegaWallFunction

3.2 Grid and Numerical Method

The geometry and mesh generation were carried out entirely within the OpenFOAM environment using a multi-stage meshing workflow. The schematic representation of the generated computational mesh is shown in Fig. 2(a). The computational domain was discretized using the *blockMesh* utility by decomposing the enclosure into 41 structured hexahedral blocks. To accurately capture the inlet jet development and near-wall flow behavior, additional local refinement was performed using *snappyHexMesh*. Fig. 2(b) shows the mesh refined near the inlet region to resolve the high velocity gradients and shear layers generated by the supply jet. Additionally, Inflation layers were also added near the walls to improve near-wall flow resolution and maintain lower $y+$ values (< 5) required by the SST k–ω turbulence model.

a)



b)

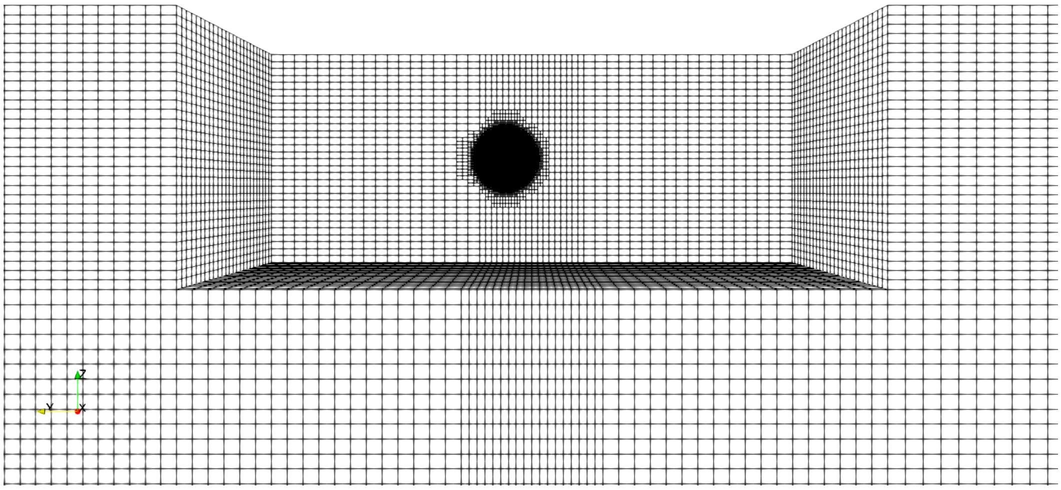


Fig. 2: (a) Grid mesh, and (b) refinement near the inlet

The numerical simulations in this study were carried out using *buoyantBoussinesqSimpleFoam*, a steady-state solver available in OpenFOAM. This solver is specifically designed for incompressible, turbulent flows where buoyancy effects are important. It is well suited for indoor ventilation problems, where temperature differences cause natural air movement. The linear solvers and numerical schemes used in this study are listed below:

Table. 2. Numerical schemes and solver settings used for the OpenFOAM simulation.

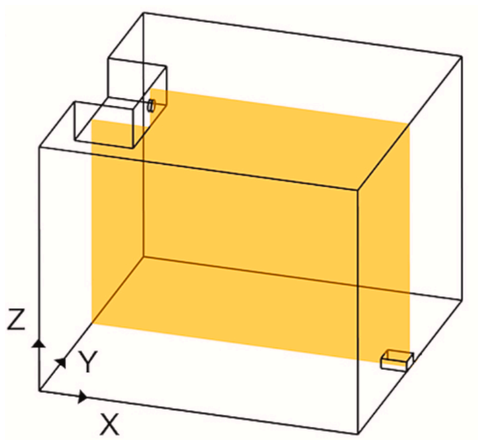
Solver	buoyantBoussinesqSimpleFoam
Algorithm	SIMPLE
Convective Scheme	linearUpwind
Turbulence Scheme	upwind
Diffusion Scheme	Gauss linear corrected
Pressure Solver	GAMG
Other Solvers	PBiCGStab + DILU

To ensure the reliability of the steady-state solutions, numerical convergence was monitored through the scaled residuals of the governing equations. The simulations were continued until all residuals specifically for pressure (p_rgh), velocity (U), temperature (T), and turbulence quantities (k and ω) reached a steady state below a threshold of 10^{-5}.

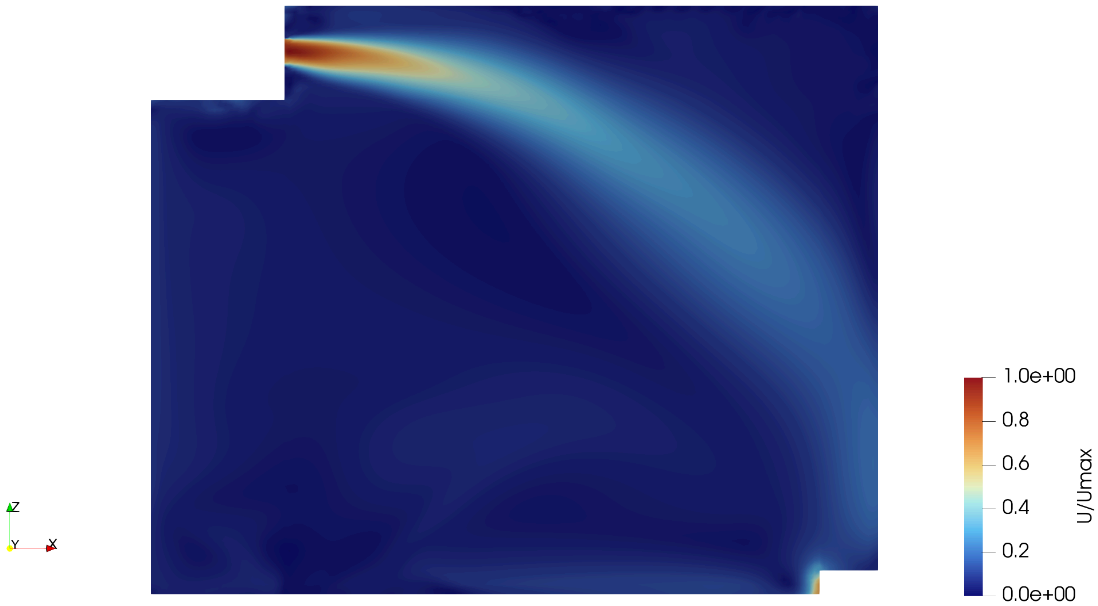
3.3 Validation

The OpenFOAM simulation is validated by comparing the numerical results with the experimental data of Gresse *et al.* [2] and ANSYS Fluent results reported by Kang and Van Hooff [1]. A vertical slice taken through the middle of the enclosure along the *y*-direction (y = 1.55 m) was used for post-processing and visualization. The location of the extracted plane is shown in Fig. 3(a). Figures 3 (b) and (c) present the contours of the velocity and air temperature profiles obtained from the OpenFOAM simulation. The velocity contour shows the development of the inlet jet as it travels downward into the enclosure due to the combined effects of inlet momentum and buoyancy. Gradual spreading and decay of the jet can also be observed as the flow moves away from the inlet region. The overall jet trajectory and recirculation behavior predicted by the simulation are close to the reference experimental data. The temperature contour indicates the presence of thermal stratification inside the enclosure. Lower temperature regions are concentrated near the inlet jet, while comparatively warmer regions are observed away from the jet core. The thermal field predicted using the prescribed inlet conditions follows the expected non-isothermal airflow behavior reported in the experimental study [2]. Minor deviations were observed near regions with strong velocity gradients and turbulent mixing. These differences are mainly attributed to mesh resolution limitations and the assumptions associated with steady RANS turbulence modeling.

a)



b)



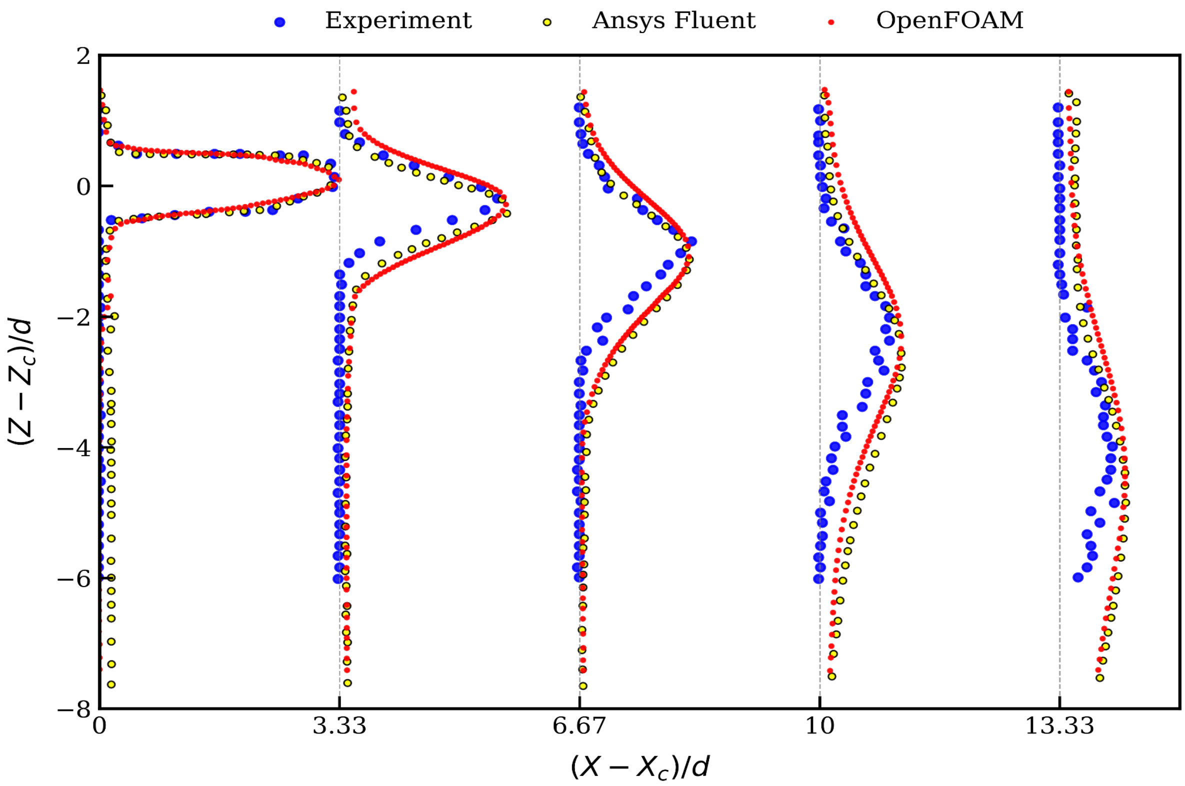
c)

Fig. 3. Contour of the CFD simulation results for the reference case. (a) contour location; (b) dimensionless velocity magnitude; and (c) dimensionless air temperature.

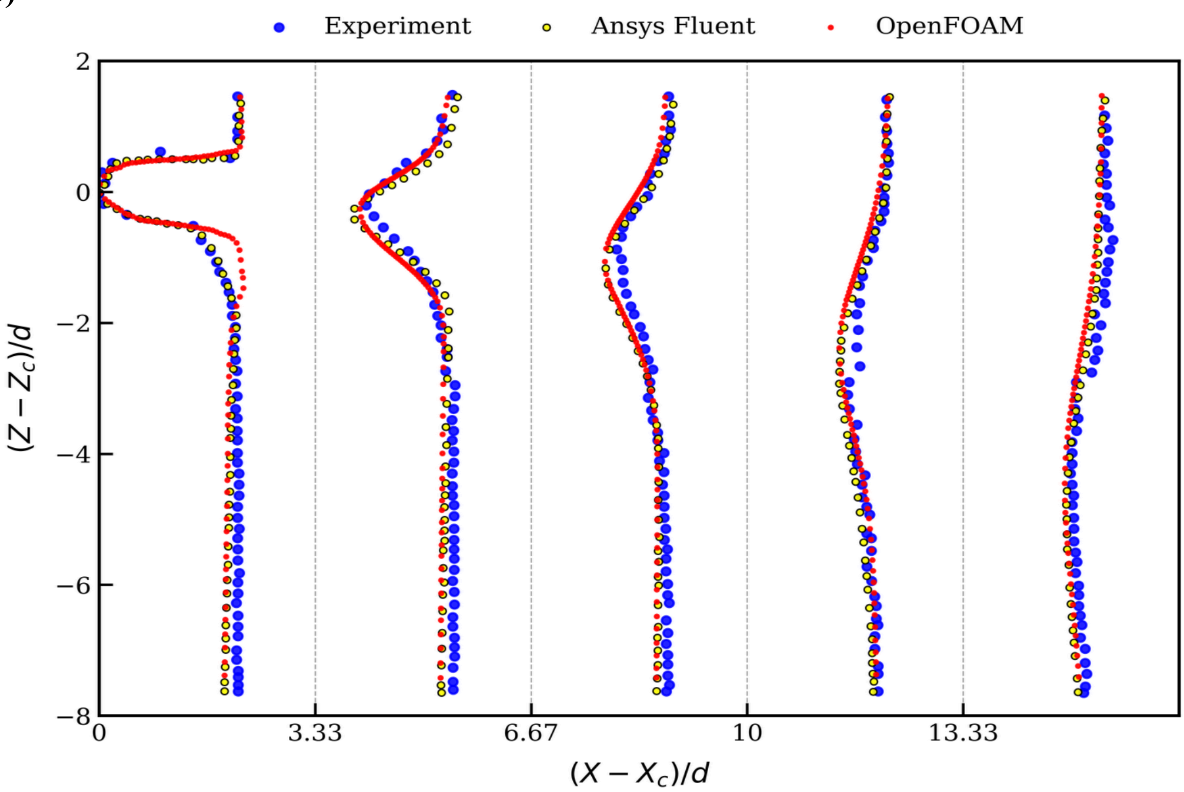
The comparison charts for dimensionless velocity magnitude, dimensionless air temperature, and dimensionless turbulent kinetic energy are presented below in Fig. 4 (a), (b) and (c) respectively. The plots compare the results obtained from the experiments [2], ANSYS Fluent [1] simulations reported in the reference study, and the present OpenFOAM simulation. Overall, the OpenFOAM results show good agreement with the experimental and ANSYS

Fluent data, with a maximum deviation of approximately 15% observed in regions with strong velocity gradients and turbulent mixing.

a)



b)



c)

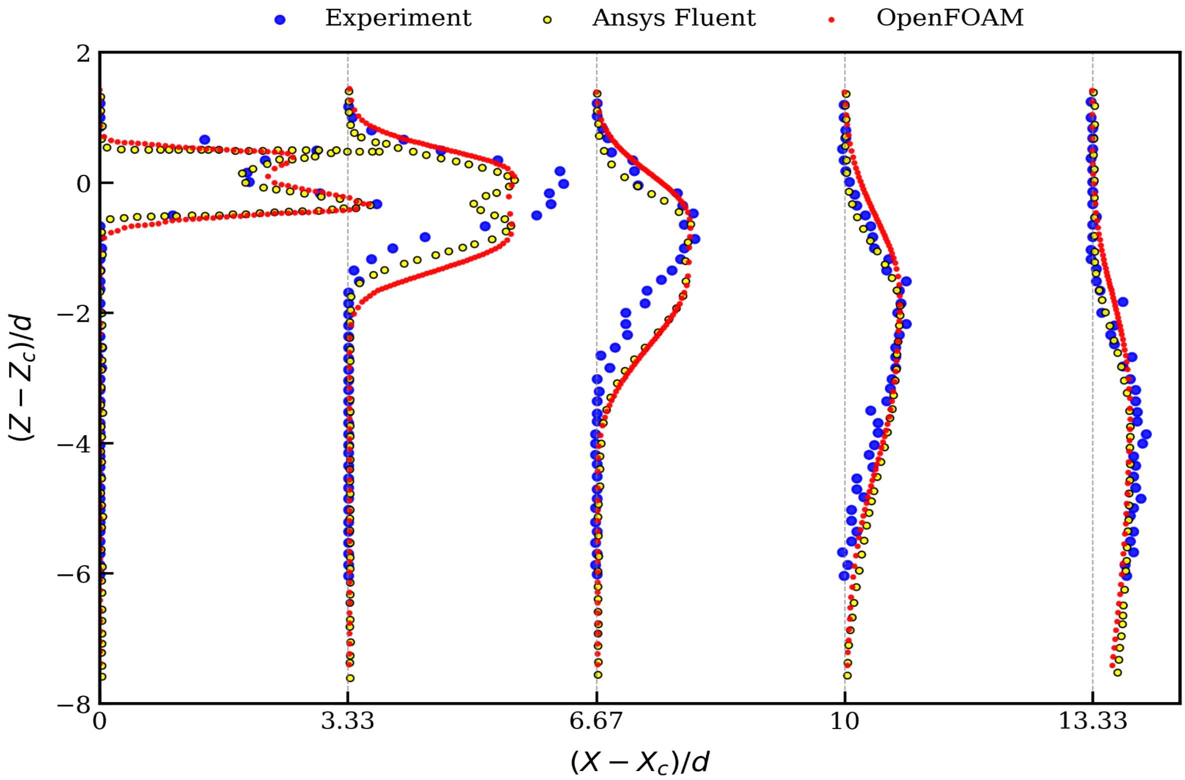


Fig. 4: Comparison of OpenFOAM results with literature for (a) Dimensionless velocity magnitude; (b) Dimensionless air temperature magnitude; and (c) Dimensionless TKE magnitude.

4. CONCLUSIONS

This work successfully developed and validated an OpenFOAM implementation of a benchmark non-isothermal mechanical ventilation case using the *buoyantBoussinesqSimpleFoam* solver and the SST k–ω turbulence model. The mesh was generated using *blockMesh* and *snappyHexMesh*, while experimentally measured inlet data were implemented using prescribed boundary conditions. The OpenFOAM simulation results for velocity, air temperature, and turbulent kinetic energy matches with the reference experimental data. The developed OpenFOAM case provides a reproducible framework for future indoor ventilation CFD studies.

REFERENCES

1. Luyang Kang and Twan van Hooff, "Influence of inlet boundary conditions on 3D steady RANS simulations of non-isothermal mechanical ventilation in a generic enclosure," International Journal of Thermal Sciences, vol. 182, 2022, Art. no. 107792, ISSN: 1290-0729.
2. Teddy Gresse, Lucie Merlier, and Frédéric Kuznik, "Detailed airflow dynamics and temperature data of axisymmetric and anisothermal jets developing in a room," Data in Brief, vol. 29, 2020, Art. no. 105382, ISSN: 2352-3409.

APPENDIX

Implementation of *timeVaryingMappedFixedValue* Boundary Condition in OpenFOAM

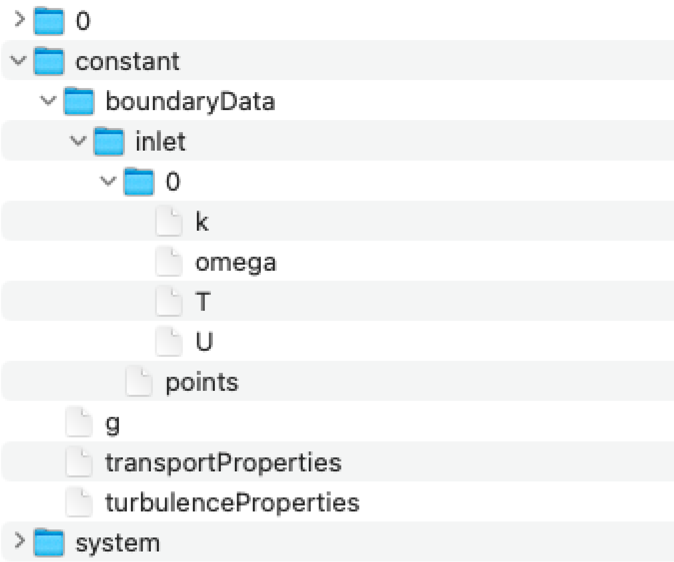
The experimentally measured inlet velocity, temperature, and turbulence quantities were implemented using the *timeVaryingMappedFixedValue* boundary condition (Prescribed Values) available in OpenFOAM. This boundary condition allows spatially varying experimental data to be mapped directly onto the inlet patch during the simulation. The inlet experimental datasets were stored inside the constant/boundaryData/inlet/directory of the OpenFOAM case.

The implementation procedure includes:

- defining the inlet sampling locations using the points file,
- storing velocity data as vector quantities,
- storing temperature and turbulence quantities as scalar field values,
- and applying the mapped boundary condition in the respective field files (U, T, k, and omega).

The same boundary condition structure was applied for all inlet quantities. This implementation allowed the CFD simulation to accurately reproduce the experimentally observed inlet jet structure, including the non-uniform velocity distribution and inlet asymmetry. As a result, improved prediction of jet development, thermal stratification, and turbulence distribution inside the enclosure was achieved. The folder structure, sample points file, inlet data format, and boundary condition implementation used in the present work are shown in Fig. A1.

a)



b)

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  v2412                                 |
|   \\  /    A nd           | Website:  www.openfoam.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/

FoamFile
{
    version     2.0;
    format      ascii;
    class       vectorField;
    object      points;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

31
(
    (0.57 1.61037 2.30535)
    (0.57 1.59058 2.33951)
    (0.57 1.59063 2.31931)
```

c)

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  v2412                                 |
|   \\  /    A nd           | Website:  www.openfoam.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/

FoamFile
{
    version     2.0;
    format      ascii;
    class       vectorField;
    object      U;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

31
(
    (0.85 -0.21 0.11)
    (0.91 0.16 0.14)
    (1.08 -0.19 0.01)
```

d)

```
boundaryField
{
    inlet
    {
        type            timeVaryingMappedFixedValue;
        fileName        "constant/boundaryData/inlet";

        setAverage      false;
        mapMethod       planar;
        outOfBounds     clamp;

        offset          (0 0 0);

        value           uniform (1.121 -0.076 -0.103);
    }
```

Fig. A1: a) boundaryData folder structure, b) example format of the points file, c) example format of the velocity vector file, and d) Boundary condition implementation in the 0/U field file.

DISCLAIMER: This project reproduces the results from an existing work, which has been acknowledged in the report. Any query related to the original work should not be directed to the contributor of this project.