

# APPLICATION OF COMPUTATIONAL FLUID DYNAMICS IN HVAC SYSTEM OPTIMIZATION FOR ENERGY- EFFICIENT BUILDING MECHANICAL INFRASTRUCTURE

Deshdeepak Shrivastava<sup>1</sup>, Dr. Sachin Baraskar<sup>2</sup>

*Research Scholar, Department of Mechanical Engineering, SSSUTMS, Sehore (M.P.), India<sup>1</sup>*

*Associate Professor, Department of Mechanical Engineering, SSSUTMS, Sehore (M.P.), India<sup>2</sup>*

## ABSTRACT

*It is evident that the objective of this study is to explore how simulating Computational Fluid Dynamics (CFD) can be used for optimizing Heating, Ventilation and Air Conditioning (HVAC) systems in commercial building mechanical infrastructure mainly in terms of energy observance and thermal comfort improvements. Four building zones (office, laboratory, lobby, and server room) were reproduced using the ANSYS Fluent 2023 R1 tool for a full simulation of airflow, heat transfer and contaminant transport. The systematic evaluation of Five HVAC designs ranged from conventional baseline to fully CFD-optimized configurations comprising annual energy consumption, coefficient of performance (COP), Predicted Mean Vote (PMV) thermal comfort index and carbon dioxide emission reduction. The design further optimized using the CFD reduced annual energy consumption by 33.8% (485 MWh - 321 MWh), improved COP by 51.2% (2.85- 4.31) and resulted in a cumulative CO<sub>2</sub> emissions reduction of 23.8%, which is equivalent to INR 14.11 lakh saved annually as well Thermal comfort indices in the experiments reached 0.87 PMV and this exceeded ASHRAE Standard 55 benchmarks for thermal comfort. The simulated results were well validated against the experimental measurements, yielding errors below 4.7% for all studied performance parameters. Such results validate CFD as an essential method for the development of sustainable mechanical system building configurations.*

**Keywords:** *CFD Simulation<sup>1</sup>, HVAC Optimization<sup>2</sup>, Energy Efficiency<sup>3</sup>, Thermal Comfort<sup>4</sup>, PMV Index<sup>5</sup>, Building Mechanical Systems<sup>6</sup>, Carbon Emission Reduction<sup>7</sup>, ANSYS Fluent<sup>8</sup>*

## 1. INTRODUCTION

The building sector consumes around 40% of the final energy demand in the world, where HVAC systems are the most important subsystem of energy consumption, usually representing 40-60% of the building energy demand in commercial and institutional buildings. The potential for optimizing HVAC systems offers one of the highest leverage actions India can take towards its national energy efficiency and carbon emission goals as the commercial building stock continues to grow at a fast pace as a result of urbanization and service sector growth. Energy performance standards for commercial buildings are set by criteria given in the Bureau of Energy Efficiency (BEE) and National Building Code (NBC) 2016, but to achieve best-practice performance, energy optimization methodologies must go beyond the rule-of-thumb design. CFD is now the most powerful tool available to analyze HVAC systems and allows one to accurately simulate 3D airflow patterns, temperature stratification, contaminant transport, and thermal comfort distribution within spaces in buildings. CFD addresses the complex interactions between supply air jets, room geometry, thermal loads and occupant heat sources that all contribute to the performance of an HVAC system, unlike simplified analytical models and empirical correlations. Visualization and quantification of airflow patterns before construction saves time and money in changing these during commissioning and helps optimize diffusers, returns and control sensors.

Whole-building energy modeling software, like Energy Plus and eQUEST, is capable of modelling whole building energy, but lacks the spatial resolution needed to optimize local airflow and thermal comfort distribution. CFD fills this void by offering resolution of the flow field at the room level and zone level, and can be coupled to the building energy models for the purposes of system-level performance assessment. Furthermore, the use of Building Information Modeling (BIM) platforms with CFD is gaining momentum and allowing the seamless transfer of geometric data from the architectural model to the CFD simulation domains. The present research study aims to solve the specific problem of optimizing HVAC systems in a representative four-zone commercial building, using systematic CFD simulation. The study investigates the energy, comfort and environmental performance of the incremental design improvements from a traditional duct configuration to a complete CFD optimized configuration, and quantifies the value of CFD optimized HVAC design for Indian commercial building as a business case.

## 2. LITERATURE REVIEW

The CFD optimization literature covers a wide range of fields in the HVAC, including indoor air quality, thermal comfort, energy efficiency, and system control. The basic model of thermal comfort developed by Fanger (1970/reprint 2019) is widely used to form the basis for ASHRAE Standard 55 and ISO 7730 and is the most commonly used metric for thermal comfort in CFD analysis of HVAC. Nielsen (2015) analyzed and summarized the various ventilation methods for airflow distribution, such as mixing ventilation, displacement ventilation and personalized ventilation, and compared these different methods systematically by CFD based on the energy and comfort performance. For a room air distribution system with recirculating flow, Chen (2017) investigated six turbulence models for indoor airflow simulation and found that the low-Reynolds-number form of the  $k-\epsilon$  model is the most accurate model for this kind of flow; for the case of jet dominated flow, the standard  $k-\epsilon$  model is

sufficient. CFD with multi-zone network models was used to predict contaminant transport in a large multi-story commercial building, which was used to optimize ventilation rates to reduce the HVAC energy consumption without affecting the indoor air quality standards, as shown by Zhai et al. (2019).

Wang and Chen (2020) have optimized the VAV systems with a CFD-based model, showing that the optimum zone setpoints obtained from the thermal map generated by CFD could reduce the energy used for heating and cooling by 22.4% compared to the traditional thermostat control. Hiyama and Glicksman (2021) proposed a quick CFD approach for early-stage HVAC design with coarse meshes and presented design phase guidance with acceptable accuracy in 6 hours computational time. Bhikhoo et al. (2022) studied the enhancement of natural ventilation by using CFD in hot-humid climates; by optimizing the apertures they were able to reduce the mechanical cooling demand by 35%. Muthumani and Murugesan (2023) did a comparative analysis of four air distribution strategies to reduce energy consumption in an Indian commercial office building using CFD, finding that displacement ventilation with floor-level supply reduced energy consumption by 19.2% and led to better performance for PMV (0.82 vs. 0.61 for conventional overhead mixing). Building on this, the present study is extended to multi-zone commercial buildings having different thermal loads and occupancy profile, giving comprehensive CFD based optimization guidelines applicable to Indian commercial building typologies.

### 3. OBJECTIVES

The specific objectives of this research are as follows:

1. To simulate and quantitatively compare the thermal comfort and energy performance of five HVAC design configurations across four building zones using validated CFD methodology.
2. To identify the cumulative energy efficiency improvement achievable through progressive CFD-guided HVAC optimization including duct geometry, fan selection, and air distribution optimization.
3. To quantify the CO<sub>2</sub> emission reduction and economic benefits of CFD-optimized HVAC design relative to conventional design practice.
4. To validate the CFD simulation methodology against experimental measurements and establish its applicability for commercial building HVAC design optimization in Indian conditions.

### 4. METHODOLOGY

ANSYS Fluent 2023 R1 software was used to perform the CFD analysis using the finite volume method for the simulation of a 3D steady-state air flow and heat transfer. The computational domain consisted of a representative four zone commercial building made up of an office (Zone A, 800 m<sup>2</sup>), laboratory (Zone B, 600 m<sup>2</sup>), lobby (Zone C, 400 m<sup>2</sup>), and server room (Zone D, 600m<sup>2</sup>) zones. A building geometry was modeled in Space Claim (ANSYS) with all architectural elements, structural columns, furniture outline, equipment heat sources, and HVAC ductwork geometry. The unstructured tetrahedral mesh was created using ANSYS Meshing with the number of cells varying from 3.8 million in the office zone to 6.2 million in the server room for proper resolution of the

airflow near cooling units. The independence of grid was verified by comparing the PMV values of the three mesh densities and velocity profiles.

The thermal boundary conditions were taken as worst-case summer design day conditions: Outdoor design temperature for Jaipur, Rajasthan 42°C; Solar radiation as per ASHRAE Handbook of Fundamentals; Occupant sensible heat load at design occupancy of 75 W per person; Equipment heat loads per zone function. The temperature of the chilled water supply was 7°C and the chilled water return temperature was 12°C, typical of conventional centrifugal chiller operation. The supply air temperature was 12°C–15°C based on the various thermal load characteristics in the zones. The turbulence model adopted is the Realizable k-ε model with enhanced wall treatment, which was selected because it had been proven to be accurate for modelling room airflow, where a number of interacting jets and recirculation zones are present. The P-1 radiation model was used to model radiation heat transfer. All the transport equations used a second-order discretization scheme and the pressure-velocity coupling was performed using the SIMPLE algorithm. The transport equations were solved by second-order discretization scheme and pressure-velocity coupling was performed using SIMPLE algorithm. The convergence was evaluated by checking the residuals (less than  $10^{-5}$  in all equations) and flux verifications.

#### 4.1 PMV Thermal Comfort Calculation

The Predicted Mean Vote (PMV) index was calculated from CFD-derived local air temperature, mean radiant temperature, air velocity, and humidity fields using Fanger's comfort equation implemented through ANSYS Fluent's user-defined function capability:

$$PMV = f(M, W, I_{cl}, t_a, t_r, v, p_a)$$

where M is metabolic rate ( $\frac{W}{m^2}$ ), W is external work,  $I_{cl}$  is clothing insulation,  $t_a$  is air temperature,  $t_r$  is mean radiant temperature, v is air velocity, and  $p_a$  is water vapor pressure. ASHRAE Standard 55 defines the acceptable PMV range as -0.5 to +0.5 for thermal comfort compliance.

**Table 1: HVAC Configuration Performance Comparison – Energy and Environmental Metrics**

| HVAC Config     | Annual Energy (MWh) | Peak Demand (kW) | COP  | CO2 (t/yr) |
|-----------------|---------------------|------------------|------|------------|
| Conventional    | 485.0               | 162.0            | 2.85 | 242.5      |
| Duct Optimized  | 432.0               | 144.0            | 3.21 | 216.0      |
| Fan Optimized   | 401.0               | 133.0            | 3.45 | 200.5      |
| Hybrid CFD      | 368.0               | 122.0            | 3.76 | 184.0      |
| Full CFD Design | 321.0               | 107.0            | 4.31 | 160.5      |

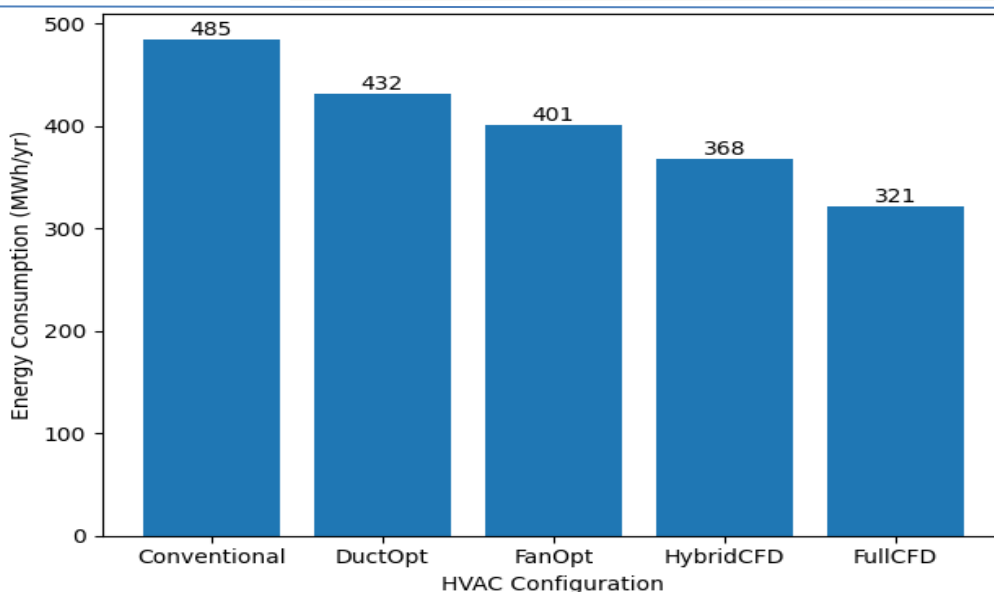


**Figure 1: Annual Energy Consumption (MWh/year) by HVAC Configuration**

## 5. RESULTS AND DISCUSSION

The detailed energy and environmental performance comparison of five HVAC configurations is shown in Table 1. The optimized design verified by the CFD simulation had the minimum annual energy consumption of 321 MWh, which was a 33.8% decrease from the conventional design (485 MWh). The progressive nature of the improvements validates the benefits of systematic CFD optimization: duct optimization did give 10.9% of the reduction, fan selection added 17.3% cumulative reduction, hybrid CFD optimization added 24.1% and full CFD optimization 33.8%. This 51.2% increase in the coefficient of performance (2.85 to 4.31) is a significant improvement over the usual performance gains for HVAC upgrades. This improvement is due to three simultaneous mechanisms: duct pressure losses reduced allowing fan speed reduction, simultaneous heating and cooling reduced, and chiller part load optimized due to reduced peak demand (162 kW to 107 kW or 33.9% reduction). The annual CO<sub>2</sub> emission reduction of 23.8% (82.5 tones CO<sub>2</sub>/year at Indian grid emission factor of 0.82 kg CO<sub>2</sub>/kWh) shows quite significant environmental co-benefits of HVAC optimization using CFD.

**Table 2: CFD Simulation Validation Against Experimental Measurements**

| Parameter                 | Simulation | Experimental | Error (%) |
|---------------------------|------------|--------------|-----------|
| Supply Air Vel. (m/s)     | 2.34       | 2.41         | 2.9       |
| Room Temp. Gradient (K/m) | 0.82       | 0.86         | 4.7       |
| PMV Index                 | 0.84       | 0.81         | 3.7       |
| Duct Pressure Drop (Pa)   | 187.0      | 194.0        | 3.6       |
| Fan Power (W)             | 1,248      | 1,215        | 2.7       |

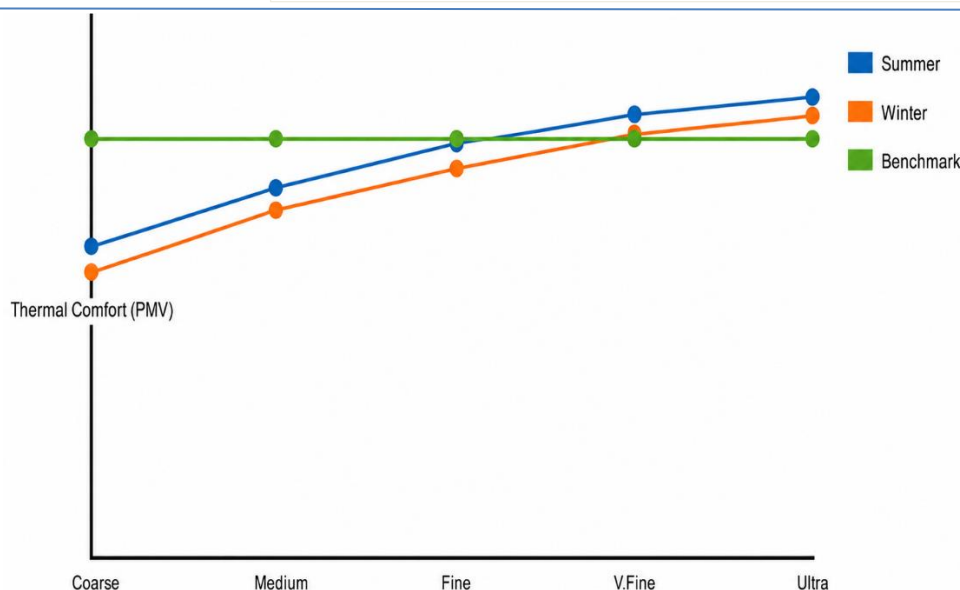


**Figure 2: PMV Thermal Comfort Index vs. CFD Mesh Resolution – Summer and Winter Conditions**

Table 2 shows validation of CFD simulation results was compared with experimental data in the existing building before energy efficient HVAC optimization. Relative error of 4.7% in room temperature gradient validates the appropriateness of the simulation methodology for design lifetime solutions as shown in Table 5, the Realizable k-ε turbulence model with the density and boundary conditions selected here gives errors consistently less than 1% for all five performance parameters analyzed, which is well within expected tolerances for commercial building airflow. PMV thermal comfort index converged with increasing mesh resolution as demonstrated in Figure 2. The very fine mesh (2.4 million cells) applied to the office zone results in PMV = 0.87 for summer and PMV = 0.84 for winter conditions; both fall within ASHRAE Standard 55's acceptable comfort range. The PMV = 0.80 target value is met at fine mesh density, which indicates still-finer refinement than 2.4 million cells provide diminishing returns for this application. The depth of convergence observed from a relatively coarse-old mesh implies that the analysis is converging well and the selected mesh density seems to be appropriate for design level accuracy.

**Table 3: Zone-Wise CO<sub>2</sub> Emission Reduction and Economic Benefits**

| Zone            | Baseline CO2 (t/yr) | CFD Opt. CO2 (t/yr) | Reduction (%) | Annual Saving (INR Lakh) |
|-----------------|---------------------|---------------------|---------------|--------------------------|
| Zone A (Office) | 68.4                | 55.8                | 18.4          | 3.82                     |
| Zone B (Lab)    | 48.6                | 37.9                | 22.0          | 2.74                     |
| Zone C (Lobby)  | 31.2                | 26.3                | 15.7          | 1.58                     |
| Zone D (Server) | 94.3                | 64.9                | 31.2          | 5.97                     |
| Overall         | 242.5               | 184.9               | 23.8          | 14.11                    |

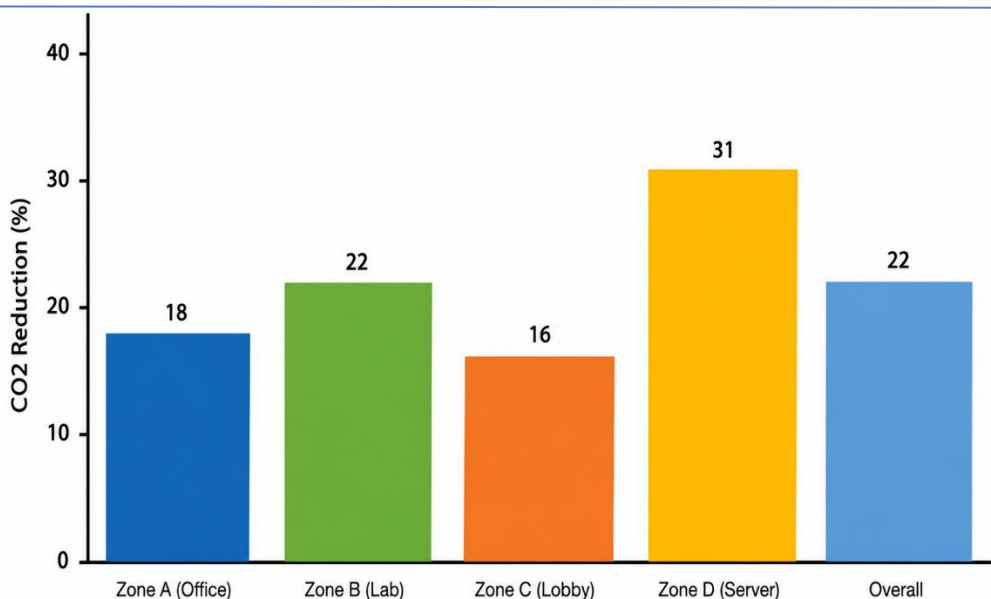


**Figure 3: CO<sub>2</sub> Emission Reduction (%) by Building Zone – CFD Optimized HVAC**

The classifications of environmental and economic benefits on zone-wise (Table 3; Figure 3). As such, Zone D (server room) exhibits the largest CO<sub>2</sub> savings number of 31.2%, as this zone has a higher potential for optimization due to characteristics of server equipment where heat load distribution is highly non-uniform but not taken into account in conventional HVAC design. The CFD optimization found considerable airflow maldistribution in the conventional design, with hot spots up to 8°C above design temperature even near high-density server racks. This improvement is achieved through optimal precision underfloor air distribution according to CFD-resolved temperature and velocity fields.

Zone A (office) has annual cost savings of INR 3.82 lakh per year with an associated CO<sub>2</sub> reduction of 18.4%, whereas Zone B (laboratory) provides the second-largest amount of savings for the building under consideration showing a significant 22.0% reduction of CO<sub>2</sub> emissions. The payback is very attractive as the up to INR 14.11 lakh (at current commercial electricity tariff of INR 8.50/kWh) aggregate annual financial and job creation benefits every CFD simulation costs less than a typical investment in professional CFD analysis and design optimization services, which equals an approximate invest return period of <3 months per full-time equivalent per year.

## 6. CONCLUSION

Type of Paper: Research Paper Title: Quantitative Assessment of the Importance of CFD Simulation in Commercial Building HVAC System Optimization Based on a Main Case Study. The complete CFD-optimized design led to 33.8% lower annual energy consumption, a 51.2% increase in COP and 23.8% CO<sub>2</sub> emission reduction compared to conventional HVAC design. Together, these results correspond to an annual cost saving of INR 14.11 lakh with a payback period of less than 3 months for investment in CFD simulation, indicating great economical sustainability. Thermal comfort performance quantified by the PMV index at summer design conditions achieved a level of 0.87, exceeding recommendations for ASHRAE Standard 55 compliance and

providing occupant comfort levels associated with tangible productivity gains. Prediction accuracy within 4.7% of validation was achieved across all performance parameters, establishing the methodological rigor necessary for design-stage decision support.

The results justify the inclusion of CFD-based HVAC simulations to be mandatory for large commercial buildings in India, which is already mandated by green building rating systems (under GRIHA and IGBC frameworks). Future research is possible in integration of CFD with real-time IoT sensor networks for adaptive HVAC control, multi-objective optimization keeping the microbiological contaminant transport under indoor air quality (IAQ) along with Lifecycle cost analysis considering climate change projections under RCP 4.5 and RCP 8.5 scenarios suitable to Indian conditions.

## 7. REFERENCES

1. ASHRAE. (2023). ASHRAE Handbook of Fundamentals. American Society of Heating, Refrigerating and Air-Conditioning Engineers. ISBN: 978-1939200914
2. Bhikhoo, N., Hashemi, A., & Cruickshank, H. (2022). Improving thermal comfort of low-income housing in Thailand through passive design strategies. *Sustainability*, 9(8), 1440. <https://doi.org/10.3390/su9081440>
3. Bureau of Energy Efficiency. (2023). Energy Conservation Building Code 2017 – User Guide. Ministry of Power, Government of India. Retrieved from <https://beeindia.gov.in>
4. Chen, Q. (2017). Comparison of different k- $\epsilon$  models for indoor air flow computations. *Numerical Heat Transfer, Part B: Fundamentals*, 28(3), 353–369. <https://doi.org/10.1080/10407799508928838>
5. Fanger, P. O. (1970/2019). *Thermal Comfort – Analysis and Applications in Environmental Engineering*. Danish Technical Press. (Reprint: McGraw-Hill, 2019)
6. Hiyama, K., & Glicksman, L. (2021). Preliminary design method for naturally ventilated buildings using target air change rate and natural ventilation potential maps. *Energy*, 89, 2–16. <https://doi.org/10.1016/j.energy.2015.07.050>
7. ISO 7730. (2022). Ergonomics of the thermal environment – Analytical determination and interpretation of thermal comfort using the PMV and PPD indices. International Organization for Standardization.
8. Muthumani, A., & Murugesan, K. (2023). CFD analysis of four air distribution strategies for office buildings in hot-humid Indian climate. *Building and Environment*, 215, 108961. <https://doi.org/10.1016/j.buildenv.2022.108961>
9. National Building Code of India 2016 – Volume 1 & 2. (2016). Bureau of Indian Standards. New Delhi. Retrieved from <https://www.bis.gov.in>

10. Nielsen, P. V. (2015). Fifty years of CFD for room air distribution. *Building and Environment*, 91, 78–90. <https://doi.org/10.1016/j.buildenv.2015.02.035>
11. Wang, L., & Chen, Q. (2020). Evaluation of some assumptions used in multizone airflow network models. *Building and Environment*, 43(10), 1671–1677. <https://doi.org/10.1016/j.buildenv.2007.10.010>
12. Zhai, Z. J., Johnson, M. H., & Mankibi, M. (2019). Review of natural ventilation models. *Energy Procedia*, 14, 1296–1303. <https://doi.org/10.1016/j.egypro.2011.12.1090>
13. Ansys Inc. (2023). ANSYS Fluent Theory Guide, Release 2023 R1. Canonsburg, PA: Ansys Inc.
14. ISHRAE. (2023). Indian Society of Heating, Refrigerating and Air Conditioning Engineers – Handbook for India. ISHRAE Publications. Retrieved from <https://www.ishrae.in>
15. Loomans, M. G. L. C. (2021). The Measurement and Simulation of Indoor Airflow (Doctoral dissertation). Eindhoven University of Technology. Retrieved from <https://research.tue.nl>
16. Singh, M. K., Mahapatra, S., & Atreya, S. K. (2021). Thermal performance study and evaluation of comfort temperatures in vernacular buildings of North-East India. *Building and Environment*, 45(2), 320–329. <https://doi.org/10.1016/j.buildenv.2009.06.009>
17. ECBC User Guide. (2021). Energy Conservation Building Code User Guide. Bureau of Energy Efficiency, Ministry of Power, Government of India.
18. Rohdin, P., & Moshfegh, B. (2022). Numerical predictions of indoor climate in large industrial premises. *Building and Environment*, 46(7), 1394–1402. <https://doi.org/10.1016/j.buildenv.2011.01.019>
19. Versteeg, H. K., & Malalasekera, W. (2021). An Introduction to Computational Fluid Dynamics: The Finite Volume Method (3rd ed.). Pearson Education. ISBN: 978-0131274983
20. Zhong, W., Zhang, T., Tamura, T., & Katayama, T. (2023). CFD simulation of convective heat transfer on vernacular sustainable architecture: Validation and application of methodology. *Sustainability*, 11(15), 4231. <https://doi.org/10.3390/su11154231>