Enhancing Thermal Comfort Analysis and Optimization of HVAC Systems Using Open-Source Software

Júlio M. Beghelli[1], Felipe P. Mariano[1]

[1]*Laboratório de Engenharia Térmica e de Fluidos, Escola de Engenharia Elétrica, Mecânica e de Computação, Universidade Federal de Goiás*
Av. Esperança, s/n, Campus Samambaia, Predio: B5, 74690-900, Goiás/Goiânia, Brazil
juliomodesto@discente.ufg.br, fpmariano@ufg.br

Abstract. Ventilation, cooling, and heating systems play a crucial role in providing thermal comfort within occupied environments, influencing productivity, well-being, health, and energy consumption. This research focuses on leveraging Computational Fluid Dynamics (CFD) as a tool for studying fluid flows and optimizing heating, ventilation, and air conditioning (HVAC) systems to achieve ideal room temperatures efficiently. Initially, a thermal comfort study is conducted using the OpenFOAM software, aiming to compare and validate its algorithm against a reference article that utilized proprietary software. By demonstrating the capabilities of OpenFOAM as an open-source alternative, this research opens opportunities for analyzing HVAC systems using an accessible program, facilitating the optimization of climatization strategies. Subsequently, an optimization analysis of ventilation systems is performed through a factorial design that involves altering the positions of air inlets and outlets, as well as adjusting the insufflation velocity. The findings reveal that superior positions of air inlets lead to improved thermal comfort results, as measured by the Air Diffusion Performance Index (ADPI). This research provides an insights into optimizing the configuration of ventilation systems to enhance occupant's thermal comfort. By utilizing CFD simulations and exploring several parameters affecting thermal comfort, this research contributes to advancing HVAC system optimization.

Keywords: Computational Fluid Dynamics, Thermal Comfort, HVAC Systems, OpenFOAM.

1 Introduction

The design and construction of buildings are driven by the aspiration to create environments that offer comfort and well-being to their occupants, regardless of external climatic conditions. However, the inherent variability of external factors such as temperature and humidity often exceeds the limits of human adaptability. Consequently, the management of internal conditions becomes imperative to establish a healthy and comfortable indoor environment [1]. In this pursuit, heating, ventilation, and air conditioning (HVAC) systems play a pivotal role in ensuring optimal thermal comfort for occupants.

[2] conducted an experimental exploration investigating diverse ventilation scenarios to assess their impact on air quality, cooling efficiency, and thermal comfort. Their study examined three distinct air distribution methods: stratified ventilation (SV), mixing ventilation (MV), and displacement ventilation (DV). The findings revealed nuanced differences in thermal comfort requirements and cooling efficiency among these approaches, shedding light on the pivotal role of adequate air distribution in creating a comfortable indoor environment. Additionally, the work by [3] demonstrated the practical application of ventilation concepts in modernizing commercial office HVAC systems. By integrating management sensors, exhaust air dehumidification, and advanced particle filtration, they achieved a remarkable 50% reduction in energy consumption while maintaining optimal internal comfort levels.

The term "comfort temperature" is defined by the [4] standard as a "mental condition" that expresses satisfaction with the thermal environment and is subjectively evaluated. Therefore, while the environment may be comfortable for some individuals, others may experience some discomfort. This is because thermal sensation depends on physical and physiological parameters, as well as the physiological responses of the human body to the environment. In an effort to standardize thermal comfort, [5] quantified air quality and thermal comfort using the concept of "age of air" through the Effective Draft Temperature (EDT) comfort rate, as shown in Equation 1.

$$EDT = (T_i - T_r) - 7.66(U_i - 0.15), \tag{1}$$

where EDT is the comfort rate, T_i is the local temperature at a certain point in Celsius (°C), T_r is the room setpoint temperature in Celsius (°C), and U_i is the magnitude of velocity at a certain point in meters per second (m/s). The EDT index provides a quantifiable indication of comfort at a discrete point in space by combining physiological effects, air temperature, and air movement on the human body [6]. The point is considered comfortable if the results of the EDT equation is within the range of -1.7 to +1.1, and the measured velocity at the point is less than 0.35 m/s.

The Air Diffusion Performance Index (ADPI) is a numerical index used in conjunction with EDT to evaluate the air distribution system in a space, and is defined as the number of points that satisfy the criteria within an occupied zone, as stipulated by [4], as shown in Equation 2.

$$ADPI = 100\frac{N\theta}{N}, \tag{2}$$

where $ADPI$ is the air diffusion performance index, $N\theta$ is the number of locations where EDT complies with the standard, and N is the total number of locations where measurements were taken. Furthermore, ADPI was designed as a performance index for thermal uniformity, quantifying the HVAC system's performance while supplying or distributing conditioned air at different positions [7]. Thus, ADPI represents the percentage of locations where the obtained values comply with the reference for EDT ($-1.7 < EDT < +1.1$) and air velocity ($\bar{V} < 0.35 m/s$). A value of 100% for the index indicates that the most desired situation has occurred, as the entire environment will be thermally uniform according to the standard.

This paper undertakes the mission of unraveling airflow patterns in conditioned indoor spaces, identifying zones characterized by recirculation tendencies that can compromise occupant well-being. A focal point of investigation lies in the assessment of thermal comfort, accomplished through the ADPI. The journey unfolds by employing CFD methodologies, with OpenFOAM as the designated simulation platform. Factorial design planning augments this exploration, enabling the simulation of diverse scenarios by varying air supply and exhaust configurations.

2 Mathematical Model

The mathematical model adopted in this article utilizes the equations of mass balance, linear momentum, and energy, applied to flows of Newtonian fluids and incompressible flows. And the Boussinesq approximation is employed to represent variations in specific mass, and the Reynolds-averaged Navier-Stokes (RANS) modeling is used with the $\kappa - \epsilon$ - RNG turbulence closure model:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0, \tag{3}$$

$$\frac{\partial}{\partial x_j}(\bar{u}_i\bar{u}_j) = -\frac{1}{\rho}\frac{\partial \bar{p_0}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\nu + \nu_t)\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\right] + g_i\beta\left(T - T_0\right), \tag{4}$$

$$\frac{\partial}{\partial x_j}\left(\bar{u_j}\bar{T}\right) = \frac{\partial}{\partial x_j}\left[(\alpha + \alpha_t)\frac{\partial \bar{T}}{\partial x_j}\right] + \frac{\bar{\phi}}{\rho C_p}, \tag{5}$$

The $\kappa - \epsilon$ turbulence model is a closure model with two equations of balance with viscosity. The standard model is based on the concepts presented by [8]. The equation for turbulent kinetic energy is given by:

$$\frac{D}{Dt}(\rho\kappa) = \bigtriangledown.(\rho D_\kappa \bigtriangledown_\kappa) + P - \rho\epsilon, \tag{6}$$

where κ is the turbulent kinetic energy, D_k is the effective diffusivity for k, P is the turbulent kinetic energy production rate, and ϵ is the turbulent kinetic energy dissipation rate. The turbulent kinetic energy dissipation rate is given by:

$$\frac{D}{Dt}(\rho\epsilon) = \bigtriangledown.(\rho D_\epsilon \bigtriangledown_\epsilon) + \frac{C_1\epsilon}{k}\left(P + C_3\frac{2}{3}k \bigtriangledown .u\right) - C_2\rho\frac{\epsilon^2}{k}, \tag{7}$$

where D_ϵ is the effective diffusivity for ϵ, C_1 and C_2 are coefficients of the model.

The $\kappa - \epsilon$ closure model has other versions, and in this article, the $\kappa - \epsilon - RNG$ closure model is adopted due to the implementation of the Renormalization Group (RNG) Theory. This model is suitable for complex shear flows involving room ventilation. The modification in the RNG version is related to the viscous transport term in the ϵ balance equation, where the C_2 term is no longer treated as a constant but is adopted as a dynamically natured function:

$$c_{\epsilon 2}(\vec{x}, t) = \tilde{c_{\epsilon 2}} + \frac{c_\mu \lambda^3 \left(1 - \frac{\lambda}{\lambda_o}\right)}{1 + \beta \lambda^3}, \tag{8}$$

$$\lambda = \frac{k}{\epsilon}\sqrt{2 S_{ij} S_{ij}}, \tag{9}$$

Then, the equation for turbulent viscosity is:

$$\upsilon_t = C_\mu \frac{\kappa^2}{\epsilon}, \tag{10}$$

where C_μ is a constant coefficient, and its value, along with the other constants of the $\kappa - \epsilon - RNG$ model used in this study, is:

$$c_\mu = 0.085; \sigma_k = 0.72; \sigma_\epsilon = 0.72; c_{\epsilon 1} = 1.44; c_{\epsilon 2} = 1.68; \beta = 0.012; \lambda_o = 4.38. \tag{11}$$

2.1 OpenFOAM Setup

OpenFOAM is a free, open-source computational software that provides functions and tools encompassing various fields of engineering and science, both in commercial and academic settings. It enables the resolution of complex flow problems [9]. In general, the OpenFOAM software functions by editing different files arranged in folders, which provide options for simulation parameters. These parameters can be adjusted by editing the simulation to match the desired physical model to be investigated. By modifying the files located in the folders “0”, “constant”, and “system”, it is possible to change the fluid’s physical parameters, boundary and initial conditions, turbulence models, analysis geometry, mesh configuration and refinement, simulation intervals, and more.

The solver used in all simulation is defined in the *controlDict* file, located within the *system* folder. Within the available options, the *buoyantSimpleFoam* algorithm is chosen, where the second-order finite volume method is applied to solve heat transfer flow in a steady-state regime using the “Semi-Implicit Method for Pressure-Linked Equations” (SIMPLE), which is a numerical algorithm used to solve the pressure-velocity coupling.

In the *controlDict* file, the number of iterations, data logging interval, and simulation time are also defined. Since the equations to be solved are suitable for steady-state conditions, in OpenFOAM, the variable $\Delta T = 1$ corresponds to 1 iteration and is no longer related to time. Furthermore, the *buoyantSimpleFoam* algorithm offers the option to choose from various RANS turbulence models and variable density models. Among the variable density models, the Boussinesq approximation is included. In other words, the chosen algorithm solves Equations 3, 4, and 5 using the $\kappa - \epsilon - RNG$ turbulence model (Equations 6 and 7).

For the interpolation functions required in the second-order finite volume method, the “Gauss linear” method is used for the approximations in Equations 4, 5, and 7, while the “Gauss upwind” method is used for Equation 6.

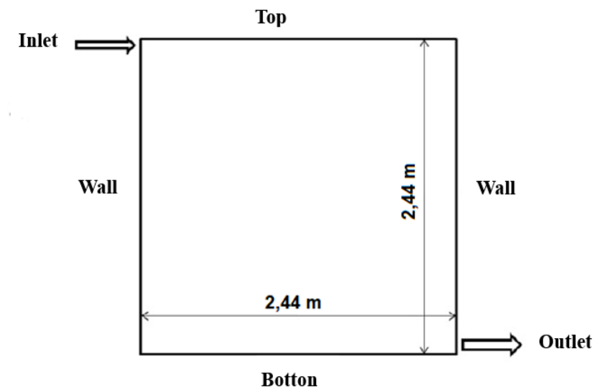
In the folder “0”, the files related to the initial and boundary conditions of all variables in the problem are located. In this folder, you can find data that pertains to both the turbulence model equations (*alphat, epsilon, k*, and *nut*) and the values of pressure (*p*), gauge pressure (*p_rgh*), temperature (*T*), and velocity (*U*). All boundary conditions are listed in Table 1.

The analyzed environment has a height and length of 2.44 m × 2.44 m. The analyzed space is two-dimensional, therefore, the depth coordinate is considered unitary for the analysis. The air inlet opening is located in the upper corner of the left wall, while the air outlet has a dimension of 8.0 cm and is located in the lower corner of the right wall, as shown in Figure 1a. This analysis space is divided into a structured and non-uniform computational mesh, as presented in Figure 1b.

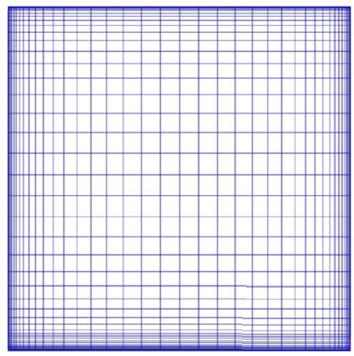
To simulate the thermally conditioned airflow in the room depicted in Figure 1 using the OpenFOAM software, it is necessary to configure the *blockMeshDict* file to define the points that represent the vertices of the geometry, as shown in Figure 1. The faces formed by the vertices of the geometry are named, thus implementing specific features for each of the 8 blocks. This allows us to determine which block represents the air inlet and outlet, as well as adjust the mesh refinement in a particular block.

Table 1. Boundary and initial conditions adopted.

Condition	U	P_{rgh}	T	κ	ϵ	ν_t	α_t
initial	(0.0,0.0,0.0)	0.0	300	0.10	0.01	0.0	0.0
inlet	(0.8,0.0,0.0)	$\partial P/\partial n = 0.0$	287.15	0.0	4.0e-6	0.0	0.0
outlet	$\partial U/\partial n = 0$	0.0	$\partial T/\partial n = 0.0$	0.0	4.0e-6	0.0	0.0
left, right, top	(0.0,0.0,0.0)	$\partial P/\partial n = 0.0$	$\partial T/\partial n = 950.57$	0.0	4.0e-6	0.0	0.0
botton	(0.0,0.0,0.0)	$\partial P/\partial n = 0.0$	$\partial T/\partial n = 2281.37$	0.0	4.0e-6	0.0	0.0
front, back	empty	empty	empty	empty	empty	empty	empty



(a) room geometry



(b) mesh with 44 x 44 non-uniform volumes

Figure 1. The two-dimensional room model used in the numerical study generated by the OpenFOAM blockMesh algorithm.

3 Results and Discussion

In order to evaluate the performance of a ventilation and air conditioning (VAC) system, two sets of simulations are proposed. Firstly, to understand the computational characteristics of the OpenFOAM software, the same problem proposed in [10] is solved. They used the FLUENT ANSYS software to simulate the flow of cold air provided by an air conditioner in a two-dimensional space. This flow is studied by varying the temperature of the supplied air, the supply velocity, and the thermal load of the room. As a result, thermal comfort is analyzed based on the ADPI, Eq. 2.

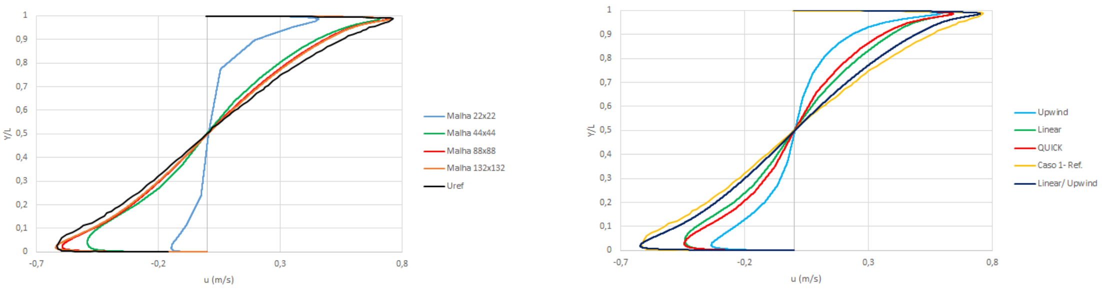
3.1 Validation of the computational parameters to be used in OpenFOAM.

In order to compare the results obtained by the OpenFOAM solver with the data presented by [10] two analyses are conducted, the first being mesh refinement and the second involving the modification of available interpolation functions. The simulations employed the following parameters, Temperature of air inlet 14 ^{o}C; Velocity of air inlet 0.8 m/s; Inlet length of air 8.0 cm; Voumetric flow inlet 24.0 L/s; and the Temperature setup 24.0 ^{o}C. Figures 2 present comparisons between different meshes for horizontal velocity profiles obtained at x/L=0.5. The profile labeled as *Ref* is the reference profile taken from the work of [10].

In Figure 2(a), it can be viewed that the more refined the mesh, the closer the results will be to the reference. Therefore, despite the simulation by [10] being performed with a mesh of 44 $\times$ 44 volumes, OpenFOAM requires a more refined mesh to obtain more accurate results. In the Figure 2(b) is shown the comparison between different interpolation functions applied in the HVAC simulations, for the horizontal velocity profile at x/L=0.5.

Interpolation functions play a significant role in obtaining results, while mesh refinement improves the results in regions close to the walls. From Figure 2, it can be observed that using two different interpolation functions for different properties is a viable alternative in OpenFOAM.

Finally, a comparison was made between the ADPI values resulting from the simulation and the values provided by [10], that the ADPI is 65.0%. In the present work the ADPI obtained is 66.0%.



(a) Mesh refinement (b) Interpolation function

Figure 2. Vertical profile of horizontal velocity, obtained at x/L=0.5.

3.2 Analysis of the systems of ventilation using factorial design

The goal is to find a combination of factors in which the cooling process fulfills its function of promoting thermal comfort efficiently *i.e.* to determine which of the configurations has the highest value of the ADPI. The configurations are determined through a 3^3 factorial design in which the factors of air inlet position, exhaust position, and air supply velocity are combined, as indicated in Table 2.

Table 2. Parameters for the 3^3 factorial design.

Inlet position	**Outlet position**	**Air supply velocity (m/s)**
Bottom	Bottom	0.5
Center	Center	0.8
Top	Top	1.2

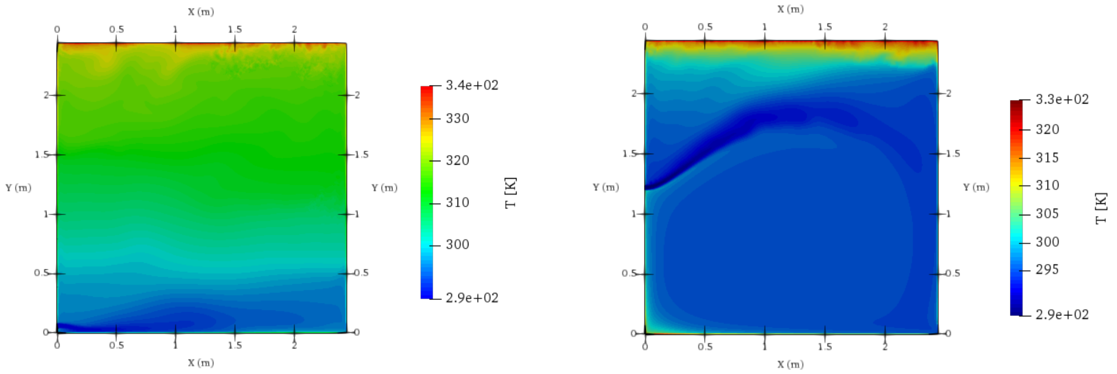
The simulated environment has the same size as presented in the previous section. The dimensions of the air inlet and outlet were kept the same in all simulated cases, with the air inlet opening measuring 4.0 cm, while the outlet opening has a height of 8.0 cm. Figure 3 shows the positions defined as“bottom”, “center” and “top”, with red arrows indicating the inlet and green arrows representing the outlet. The mesh division is not shown in Figure 3, and it should be clear that for each case in the factorial design, only one “inlet and outlet” pair is enabled.

Figure 3. Air inlet and outlet positions.

The generated mesh is structured and non-uniform, consisting of 300 $\times$ 300 volumes, and the algorithm used is *buoyantSimpleFoam* with one million iterations. All other physical and computational parameters are the same as established in preview section and are kept unchanged. The focus of the thermal comfort analysis lies in the variation of the air inlet and outlet positions for different air supply velocities. After conducting the 27 simulations, the EDT index is calculated for the entire domain, and values corresponding to $y = 1.8$ m are extracted. With this data, the ADPI is calculated for each simulation.

We will present only three cases, the worst, the best and an intermediated scenarios. For cases where air enters through the bottom of the environment and is extracted from both the bottom section and the center section, the ADPI results indicated a situation with the lowest possible ADPI. In the Figure 4(a) is shown that the temperature distribution in the environment for case at both, inlet and outlet, are at the bottom.

According to Figure 4(a), it can be observed that despite the supply of cold air into the environment, the area representing the sensation of thermal comfort, as per the standard, has ambient temperatures above the values



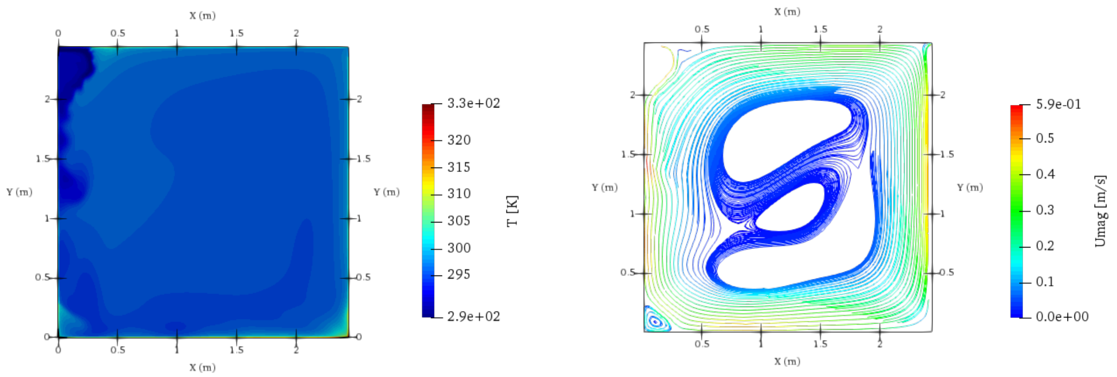
(a) Air supply both inlet and outlet at the bottom (b) Air supply both inlet and outlet at the center

Figure 4. Temperature field.

required by [4]. Thus, the result of the EDT exceeds the permissible limit at all analysis points. Therefore, the ADPI of the environment is considered as 0.0%. Despite the variation in velocity, the ADPI does not improve, indicating that for these cases, the positioning of air supply and exhaust were detrimental to the thermal comfort outcome.

In the Figure 4(b), which comprise the case where air intake is performed through the center of the room, with lower intake velocity (0.5 m/s), it can be concluded that the magnitude of the velocity was insufficient to promote the necessary air circulation to cool the environment. By increasing the intake velocity in the configurations with middle air intake, the ADPI result indicated data above the range of 13%. Although the calculated ADPI value is below 80%, the increase in velocity showed that, for these conditions, velocity has an impact on the outcome, unlike the variation in the position of the air exhaust.

The group of simulations that involve both air supply and exhaust at the top achieved the best ADPI results the resulted in an ADPI value of 100%, meaning that the EDT data obtained are in agreement with the limits imposed by [4]. In the temperature field in Figure 5. It can be observed that the entire environment is cooled, and the area with the lowest temperature is the wall near the inlet. Since the air inlet velocity is relatively low, the colder air does not remain in the EDT calculation zone and therefore does not directly affect the result. Due to heat exchanges, the environment is cooled to a temperature close to the comfort temperature.



(a) Temperature (b) Streamlines

Figure 5. Simulations with air supply and exhaust at the top.

The streamline patterns, color-coded by velocity, exhibit three zones of recirculation, as shown in Figure 5. The uppermost recirculation area corresponds to the calculation zone of the thermal comfort index. To have an impact on the result, the velocity magnitude should be greater than 0.35 m/s, which is not the case for this simulation. Therefore, these recirculations do not affect the EDT and the ADPI.

4 Conclusions

Simulations of airflow generated by air conditioning systems were conducted. In a specific environment, the air supply velocity and the positioning of air inlets and outlets were varied. Thus, the impact of different configurations on thermal comfort was investigated using the concepts of EDT and ADPI.

It was observed that OpenFOAM is capable of providing results similar to proprietary platforms for problems involving heat transfer in flows. However, the user is required to study mesh refinement, equation compatibility, interpolation functions, and adjust constants and turbulence models for each simulated problem. Factorial design enables comparative analysis of airflow behavior with heat transfer. The results confirmed the rationale for installing cooling systems at the top of the environment, as the simulations that met the requirements of [4] for thermal comfort were those with air supply positioned in the upper portion of the room. The importance of installing air outlets or exhaust fans, which contribute to proper air renewal in the environment, was also emphasized. However, they must be properly positioned, and air supply velocities must be adjusted to ensure thermal comfort.

Acknowledgements. The authors would like to thank FURNAS Centrais Elétricas, the Research and Technological Development Program (P&D) of ANEEL and the Universidade Federal de Goiás (UFG) and Graduate Program in Mechanical Engineering (PPGMEC-UFG) for supporting the development of this work.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] W. F. Stoecker and J. W. Jones. *Refrigeração e ar condicionado*. McGraw-Hill do Brasil, 1985.

[2] Y. Cheng and Z. Lin. Experimental study of airflow characteristics of stratum ventilation in a multi–occupant room with comparison to mixing ventilation and displacement ventilation. *Indoor Air*, , n. 25, pp. 662–671, 2015.

[3] W. W. Che and others. Energy consumption, indoor thermal comfort and air quality in a commercial office with retrofitted heat, ventilation and air conditioning (HVAC) system. *Energy & Buildings*, , n. 201, pp. 202–215, 2019.

[4] ASHRAE. *ASHRAE STANDARD 55. Thermal Environmental Conditions for Human Occupancy*, 2013.

[5] A. Limane and others. Three-dimensional OpenFOAM simulation to evaluate the thermal comfort of occupants, indoor air quality and heat losses inside an indoor swimming pool. *Energy & Buildings*, , n. 167, pp. 49–68, 2018.

[6] SIMSCALE. Tutorial: Thermal Comfort Parameters for HVAC Simulations, 2016.

[7] S. Samiuddin and I. Budaiwi. Assessment of thermal comfort in high-occupancy spaces with relevance to air distribution schemes: A case study of mosques. *Building Services Engineering Research & Technology*, , n. 39, pp. 572–589, 2018.

[8] B. Launder and D. Spalding. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*, vol. 3, n. 2, pp. 269–289, 1974.

[9] OpenFOAM. About OpenFOAM, 2021.

[10] A. A. Youssef. Studying comfort in a room with cold air system using computational fluid dynamics. *Ain Shams Engineering Journal*, , n. 9, pp. 1753–1762, 2018.