



Optimization of the Air Conditioning System for an Unoccupied Office Building using Split Unit and Double-Panel Radiator

Nurhayati Baharudin¹, Bukhari Manshoor^{2*}, Djamal Hisein Didane², Izzuddin Zaman², Muhammad Adib Ikhmal Mazlan², Abdul Halim Sobri Saddun², Muhammad Nur Salam Naharazmi², Muhammad Adam Abu Hasan², Muhammad Faqarazi Mohd Sam², Abdul Rafeq Saleman³, Sudher Kumar⁴

¹ Faculty of Education and Technical Vocational, Universiti Tun Hussein Onn Malaysia, Parit Raja, Batu Pahat, 86400, MALAYSIA

² Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, Parit Raja, Batu Pahat, 86400, MALAYSIA

³ Faculty of Mechanical Technology and Engineering, Universiti Teknikal Malaysia Melaka, Durian Tunggal, 76100, Melaka, MALAYSIA

⁴ Institute for Regenerative Energy Technology (in.RET), Nordhausen University of Applied Sciences, Nordhausen, 99734, GERMANY

*Corresponding Author

Received 25 Aug 2024;
Accepted 14 Oct 2024;
Available online 15 Dec 2024

Abstract: Ensuring a comfortable indoor environment in office settings is crucial for maintaining worker productivity and health. This study leverages computational fluid dynamics (CFD) to analyze and optimize the air conditioning system of a mid-sized office building, addressing issues of uneven temperature distribution and energy inefficiency. By creating a detailed CAD model of the office and performing CFD simulations using the k- ϵ turbulence model, we identified specific zones with temperature inconsistencies and assessed the factors contributing to these variations. Our results indicate that relocating the air conditioning unit to the corridor wall significantly improves temperature uniformity and reduces energy consumption compared to other placements. The findings highlight the potential of CFD in enhancing HVAC system design, thereby improving occupant comfort and reducing operational costs. This study contributes to the broader goal of optimizing energy use in commercial buildings and demonstrates practical applications of CFD in real-world settings. In this project we will assume Case B is the best because the temperature at the inlet air conditioner because it does not expose to radiation make the temperature to decrease. The highest value for case B is 302.32 and the lowest go to chair 2 is 293.98.

Keywords: HVAC Optimization, Temperature Distribution, Computational Fluid Dynamics (CFD)

1. Introduction

Ensuring a comfortable indoor environment is important for maintaining employee productivity, health, and overall well-being in office settings. The efficiency of HVAC (heating, ventilation, and air conditioning) systems is important for achieve this target. Conventional HVAC design and optimization strategies are sometimes inadequate when it comes to addressing specific challenges such as inconsistent temperature distribution and excessive energy use [17]. Computational fluid dynamics (CFD) is a powerful technology that has the potential to significantly improve HVAC system

efficiency because of its capability to study and optimized temperature and airflow patterns inside the buildings. In a number of surrounding CFD has been predominantly used in designing room air conditioning systems which include such places as laboratories, classrooms and residential homes. In addition, these types of research have shown how important they are to look at the following important factors when evaluating HVAC designs air speed; temperature distribution patterns throughout its volume as well as rate of flow among others [18]. However, the application of CFD to office environments, where worker comfort and energy efficiency are paramount, remains relatively underexplored.

A mid-sized office block has been selected as the case study in this research work that is faced with a problem of not keeping the indoor temperature within the acceptable range leading to cold areas and hot spots as well as energy inefficiency concerns about the air conditioning system. This work aims at evaluating the performance of the air conditioning system in place, establishing the main sources of the problems, and coming up with solutions that will make the situation better by employing Computational Fluid Dynamics (CFD) techniques. Within the design and optimization of Heating, Ventilation, and Air Conditioning (HVAC) systems, Computational Fluid Dynamics (CFD) is currently a necessity. This has led to significant improvement in performance and understanding of HVAC systems across many building types since CFD is capable of simulating fluid flow as well as predicting thermal distribution.

This study attempts to identify different office zones that have temperature changes and evaluate the variables causing these differences by building a comprehensive CAD model of the building and doing CFD simulations. The study also assesses how well the suggested fixes work to improve thermal comfort and energy efficiency. By optimizing energy use in commercial buildings, the research's conclusions have the potential to greatly improve workplace comfort, save operating costs, and support larger sustainability goals. In high-rise office buildings with atriums, Zhao and Li [6] studied indoor air distribution using CFD simulations. Their work highlighted the importance of CFD in understanding and improving air distribution in complex building structures. In high-rise office buildings with atriums, Zhao and Li [6] studied indoor air distribution using CFD simulations. Their work highlighted the importance of CFD in understanding and improving air distribution in complex building structures [12].

Different building types such as offices, classrooms, and laboratories have seen various studies reveal the truth of Navier-Stokes equations in the application of Computational Fluid Dynamics (CFD). For instance, how different design configurations affected thermal comfort and air quality was displayed as used CFD for air distribution modeling in a classroom [5]. This HVAC design to remind about suitable learning environment needs to look at the rate of air movement, how the temperature is spread around this place, as well as its moisture content, wetness or dryness are the key point [13]. Chen [2] compared different $k-\epsilon$ turbulence models for indoor air flow computations, highlighting the model's significance in accurately simulating indoor environments [11].

Similarly, investigated the performance of a laboratory's HVAC system using CFD simulations [6]. They identified critical areas where temperature inconsistencies were prevalent and proposed modifications to improve the system's efficiency. Their work underscores the value of CFD in identifying and addressing specific issues within HVAC systems [14]. Despite the fact that CFD has been researched sufficiently in classrooms and labs, on the issue of its use in the office environment there

are not many studies that have been done. Nonetheless, on the other hand, available few studies form the basis for understanding the specific problems that might be encountered at work. Conducted a study where they looked into how HVAC systems perform in office buildings with CFD simulations for example. They also mentioned that it is necessary for temperatures to be distributed evenly and suggested changes that could increase the effectiveness of these systems as well as comfort for those inside the buildings [7].

In office environments, where worker comfort and energy efficiency are paramount, the use of CFD remains relatively underexplored. Studies such as those by Jacob and Schälén [8] have demonstrated the benefits of CFD in evaluating HVAC performance, identifying temperature inconsistencies, and proposing modifications to improve system efficiency. The enhanced focus on energy efficiency and sustainability when running buildings has called for more optimized HVAC systems. According to Awad et al., better CFD info can be used in upgrading air condition systems to realize affirmative energy changes that also meet wider objectives on sustainability [8]. Their research demonstrated that CFD-based optimizations could reduce energy consumption by up to 20%, thereby lowering operational costs and the carbon footprint of buildings [8].

Though CFD is beneficial in the design of HVAC systems, it is still challenging to apply it in such environments as offices. Research should therefore look at the way forward concerning the integration of advanced CFD techniques into real-time monitoring systems aimed at improving the precision and applicability of simulations. Also, there is a call for more studies to concentrate on what is needed for office buildings considering the different thermal loads plus occupancy patterns as other factors. While CFD has shown great promise in the design and optimization of HVAC systems, its application in office environments presents unique challenges. Mustafa et al. [8] highlighted the complexities of natural ventilation in office buildings using CFD, suggesting that further research is needed to integrate advanced CFD techniques with real-time monitoring systems for improved precision and applicability.

The drive towards energy efficiency and sustainability in building operations has underscored the need for optimized HVAC systems. Awad et al. [8] demonstrated that CFD-based optimizations could reduce energy consumption by up to 20%, thus lowering operational costs and the carbon footprint of buildings. This aligns with the findings of Hassid et al. [5], who investigated the impact of urban heat islands on air conditioning loads and emphasized the need for energy-efficient HVAC designs.

2. Methodology

Computational fluid dynamics consists of actually solving Navier-Stokes equations; or one may say his mathematical relationships which are used widely across various engineering disciplines like transportation (including cars and planes), weather forecasting as well as chemical processing plants among others. These relationships

consider among other things; amount of material being transported per unit time, change in velocity or direction with respect to time as well as changes in temperature over a given interval of spatial distance.

2.1 Governing equations

In this study, the governing equations for continuity, momentum, and energy need to be solved simultaneously [18]. The equations are the continuity equation, momentum equation and energy equation as written in the following Eqn. 1 – 6.

Continuity equation;

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (1)$$

Momentum equation:

$$\rho \left(u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = -\frac{\partial p}{\partial x} + \eta \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \quad (2)$$

$$\rho \left(\frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = -\frac{\partial p}{\partial y} + \eta \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \quad (3)$$

$$\rho \left(\frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) = -\frac{\partial p}{\partial z} + \eta \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) \quad (4)$$

Energy equation for the fluid zone:

$$\left(u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} \right) = \frac{\lambda}{\rho x^2} \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) \quad (5)$$

Energy equation for the solid zone:

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} = 0 \quad (6)$$

2.2 CAD model and meshing

The images provided are from an ANSYS simulation and illustrate the steps of CAD modelling and meshing, crucial in finite element analysis (FEA). The first image shows a meshed CAD model, where the geometry is a solid block with various internal and external features like slots, holes, and cutouts, suggesting a mechanical component requiring detailed analysis. The model is discretized into triangular or tetrahedral elements, with a relatively uniform mesh density, ensuring areas with intricate details have finer mesh elements for accuracy.

The second image offers a transparent view of the CAD model, highlighting its internal features and providing a clear visualization of its internal structure. This transparency helps in understanding the placement and interaction of internal geometrical features, which could be channels for fluid flow, spaces for electrical components, or structural supports. The scale bar indicates dimensions in millimetres, suggesting a large overall size and the coordinate axes denote the model's orientation. The detailed CAD model and its meshing imply preparation for a finite element analysis, such as structural, thermal, or fluid dynamics simulations. The meshed model is ready for simulation, with mesh quality and density being pivotal for accuracy and efficiency. The ANSYS 2024 R1 Student version watermark indicates the use of a student edition of ANSYS, which may have limitations in model size and computational capacity [19].

Next steps include refining the mesh based on initial simulation results, applying appropriate boundary conditions and loads, running the simulation to analyze part behavior under specified conditions, and post-processing to examine results like stress concentrations, temperature distributions, or fluid flow patterns [1]. This comprehensive analysis highlights the importance of CAD modeling and meshing in preparing for accurate and efficient simulations in ANSYS. The room is modelled with simple and basic elemental blocks [15]

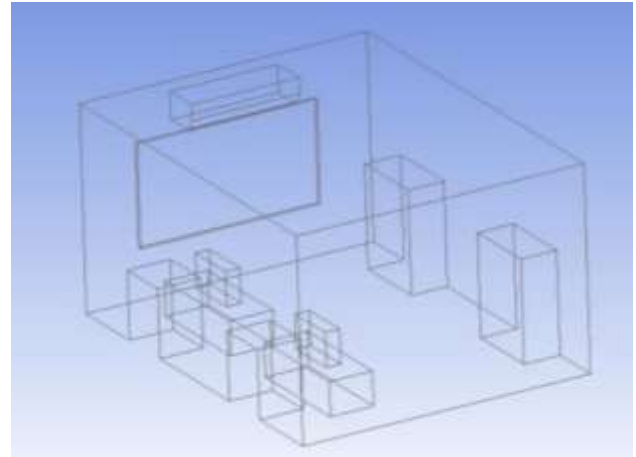


Fig. 1 - The geometry of the office room

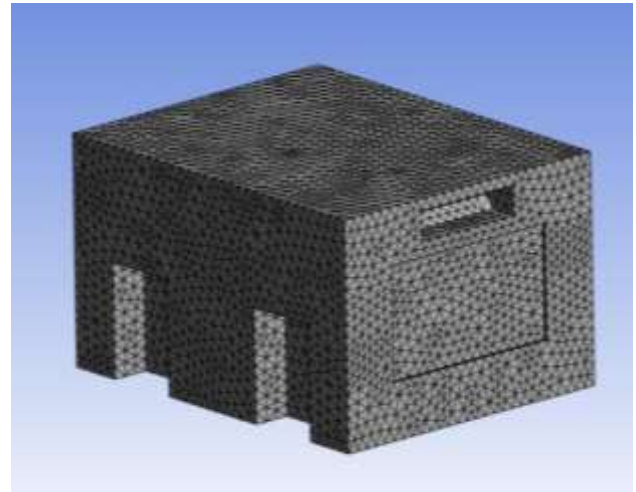


Fig. 2 - The meshing of geometry

2.3 Boundary condition

The previous study conducted experiment on an office room temperature where they tested for specified inlet temperature and measure the outlet temperature. According to the guide for big room cooling units, the air coming in moves at 1.5 m/s, and the pressure is the same as outside to match the air flow and pressure needs for big places [16]. It's 34 °C outside, and inside the room it's 25.5°C. The split room is at 30 °C, the same as the corridor outside. After checking everything, it was found that these are the right sides and heat levels needed for the room we want to cool.

The walls of the hall, not cooled or heated, add some heat to the rooms that are. But this heat is less than what comes from other areas. We thought the rooms next to the chosen one were also not heated or cooled. When it's hot or cold out, these nearby room walls share heat with the cooled or heated room. This extra heat has to be taken out by the AC system [20]. The window facing the sun gets hotter than the wall exposed to the sun. This is because glass lets heat in two ways: by letting sun rays pass through and by heat moving through it.

3. Results and Discussion

The validation of this study is done by comparing the case a result. The validation of the experiment test and simulation test for the temperature of chair1, chair2, comp1, comp2 and floor. Table 1 and 2 shows the comparison for the simulation with the previous results for Case A, B, C and D.

Table 1 - Result validation for Case A and B

Cases/ Object	Temperature (K)			
	Case A (Sim)	Case A (Ref)	Case B (Sim)	Case B (Ref)
Chair 1	298.47	308.00	295.42	300.00
Chair 2	304.51	308.00	293.98	300.00
Comp 1	302.27	312.00	302.32	319.00
Comp 2	304.46	312.00	299.99	319.00
Floor	322.27	304.00	301.11	296
Avg. Temp	306.50	307.00	298.36	305.00
% Deviation from ASHRAE Standard indoor temp. (299K)				
Deviation %	2.51%	2.85%	0.21%	2.18%

Table 2 - Result validation for Case C and D

Cases/ Object	Temperature (K)			
	Case C (Sim)	Case C (Ref)	Case D (Sim)	Case D (Ref)
Chair 1	293.13	304.00	290.58	304.00
Chair 2	297.57	304.00	292.20	304.00
Comp 1	298.78	319.00	298.34	320.00
Comp 2	299.44	319.00	301.52	320.00
Floor	301.58	304.00	309.01	304.00
Avg. Temp	318.10	308.33	298.33	305.00
% Deviation from ASHRAE Standard indoor temp. (299K)				
Deviation %	6.39%	3.29%	0.22%	3.29%

The data in Table 1 for Case A shows notable discrepancies between journal and simulation temperatures for various objects, with simulations generally predicting higher temperatures. For instance, Chair 1's temperature is 308 K in the journal but 298.47 K in the simulation, and Comp 1 shows an even larger difference, from 312 K to 302.27 K. The average temperature in the journal is 307 K, whereas the simulation average is significantly higher at 306.50 K. These discrepancies may stem from simplifications or assumptions in the simulation models, differences in environmental conditions, or unaccounted heat dissipation in real-world measurements. The findings suggest a need for more accurate thermal modeling, especially for components like computer parts where overheating is

critical and underscore the importance of validating simulations with empirical data for reliable predictions.

The comparison of temperatures for Case B between previous study and simulation data reveals differences across various objects, though less pronounced than in Case A. For Chair 1, the journal temperature is 300.00 K, while the simulation records a slightly lower temperature of 295.42 K. Chair 2 shows a similar trend, with a journal temperature of 300.00 K compared to a simulation temperature of 293.98 K. The computer components (Comp 1 and Comp 2) also exhibit differences, with journal temperatures of 319.00 K versus simulation temperatures of 302.32 K and 299.99 K, respectively. The floor's temperature in the journal is 296.00 K, compared to 300.11 K in the simulation. The average journal temperature is 305 K, slightly higher than the simulation average of 298.36 K.

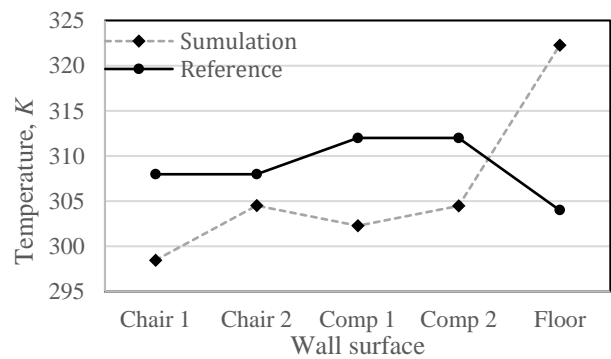


Fig. 3 - Comparison of result for validation case A

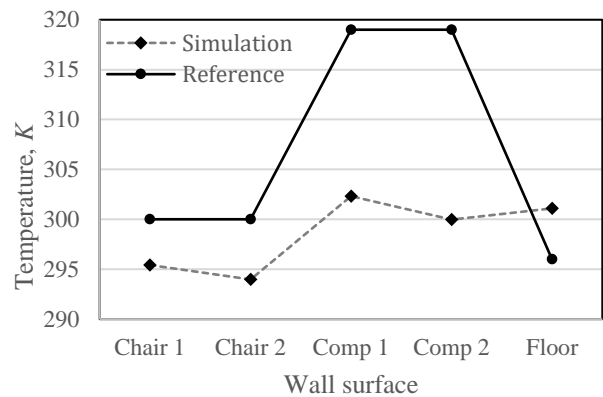


Fig. 3 – Comparison of result for validation case B

Table 2 shows the compares the temperatures of various objects measured for case C and D, both for simulation with the previous study. The recorded temperatures in the previous study show Chair 1 and Chair 2 at 304.00 K, Comp 1 and Comp 2 at 319.00 K, and the Floor at 304.00 K. In contrast, the Stimulation case reports lower temperatures for Chair 1 (293.13 K) and Chair 2 (297.57 K), and for Comp 1 (298.78 K) and Comp 2 (299.44 K), while the Floor's temperature is slightly higher at 301.58 K. The average temperature across all objects in previous study case is 307.00 K, whereas the average in the Stimulation case is significantly higher at 318.10 K. These differences highlight potential discrepancies

between observed data and simulation results, which could arise from measurement inaccuracies, assumptions in the simulation process, or variations in environmental conditions during data collection and simulation. Understanding and addressing these discrepancies is vital for validating the accuracy of simulation models.

Comparison also had been made graphically as shows in Figure 4 and 5 below. The graph highlights significant discrepancies between the observed data and the simulated data, particularly for Comp 1 and Comp 2, where the Journal temperatures are considerably higher. These differences underscore the importance of addressing potential variances in data collection methods, environmental factors, and simulation accuracy to ensure reliable simulation models.

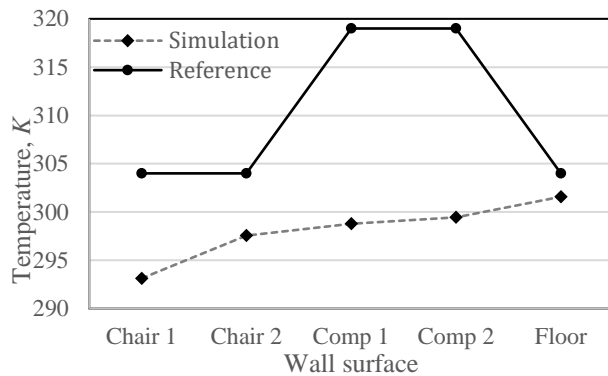


Fig. 4 – Comparison of result for validation case C

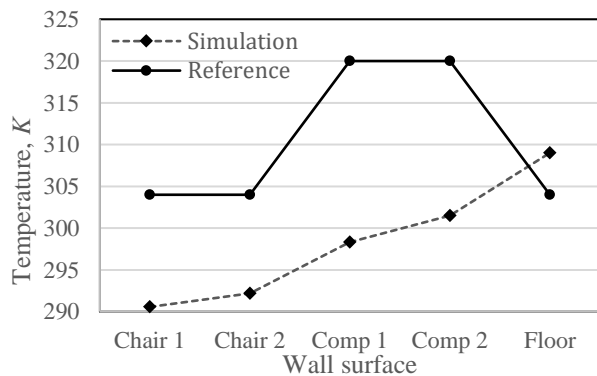


Fig. 4 – Comparison of result for validation case D

From the figures, the average temperature from the previous study is 308.33 K, while in the simulation case it is 298.33 K. These differences highlight significant discrepancies between observed and simulated data, particularly for the chairs and computers. The observed temperatures are consistently higher, indicating possible variations in environmental conditions, measurement techniques, or assumptions in the simulation process. Addressing these discrepancies is crucial for improving the accuracy and reliability of simulation models to ensure they more accurately reflect real-world conditions.

In case D, as in Figure 5, shows temperature of 304 K for both Chair 1 and Chair 2, peaks at 320 K for Comp 1 and Comp 2, and returns to 304 K for the Floor for previous study data while for the simulated results, it shows that the low temperature at 290.58 K for Chair 1, gradually increases through Chair 2 at 292.20 K, Comp 1 at 298.34 K, and Comp 2 at 301.52 K, reaching 309.01 K for the Floor. This figure reveals that the previous data temperatures are consistently higher than the simulation temperatures, with significant discrepancies for the computers, where Journal temperatures are notably higher (320 K) compared to Simulation temperatures (298.34 K and 301.52 K). The Floor temperature is an exception, with the Simulation temperature (309.01 K) being higher than the Journal temperature (304 K). These differences highlight potential variations in environmental conditions, measurement methods, or assumptions in the simulation process, emphasizing the need to address these discrepancies to improve the accuracy and reliability of simulation models.

3.1 Temperature distribution

The contour plot for temperature distribution across a plane shows in the following figures. The figures for streamlines for all cases also be an interesting result that will be highlighted here. Figure 5 shows the results for case A where the contour shows the temperature distribution in the entire room and at the cross-section of the room. From the figure, it clearly shows that the higher temperature come from the sources of heat which is at the computers area. The heat then move to the upper region in the room.

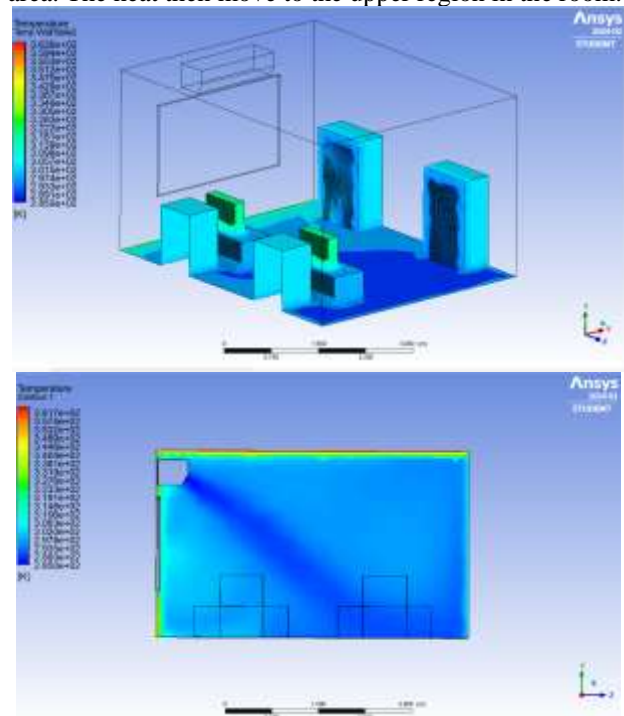


Fig. 5 – Temperature distribution for case A

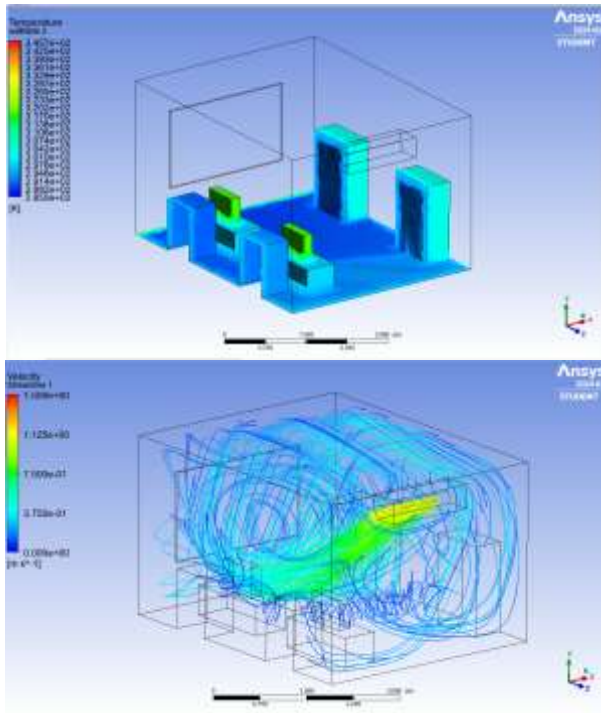


Fig. 6 – Temperature distribution and streamline for case B

Figure 6 shows the contour temperature and streamline for the room in case B. From the contour temperature we can analyze that comp 1 and comp 2 have highest temperature in case B that validate our simulation. The streamline shows the movement of the air in case B.

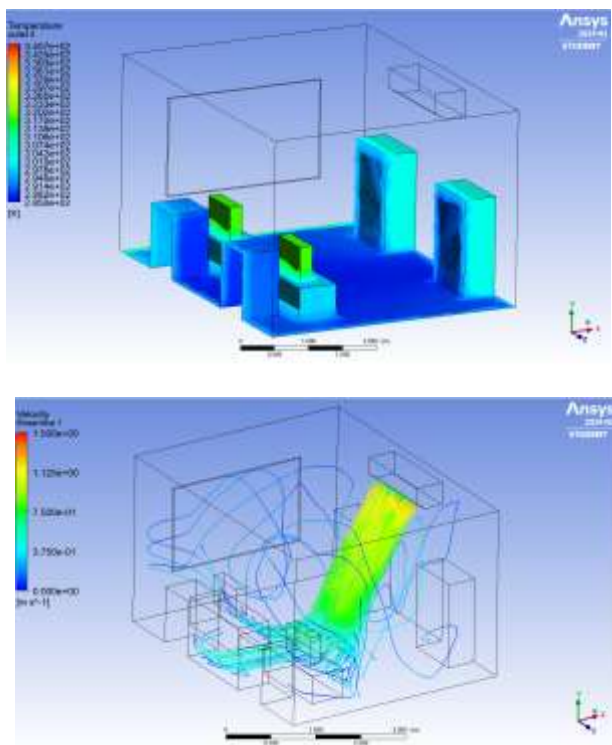


Fig. 7 – Temperature distribution and streamline for case C

Figure 7 is the results for case C while Figure 8 is the results for case D. The average temperature for case C in the stimulation gets 318.10K and it higher than the journal for case C which it gets 307K. The average temperature for case D in the stimulation gets 298.33K, it more less than case D journal which it 308.33.

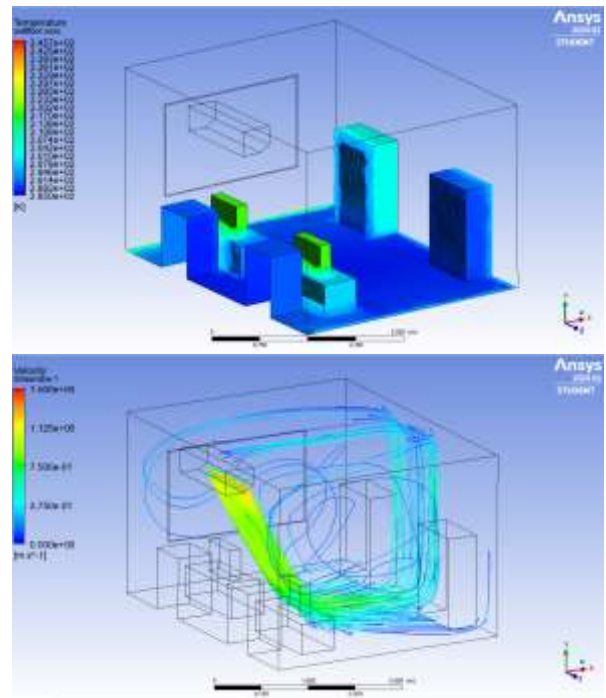


Fig. 8 – Temperature distribution and streamline for case D

4. Conclusion

In conclusion, the comparative analysis of total temperature data from both the journal and simulation for various wall surfaces reveals notable discrepancies, particularly in peak temperatures and overall distribution. The temperatures of various objects under different cases and their deviations from the ASHRAE standard indoor temperature of 299 K. The average temperatures and deviations are provided for both journal and simulation data across four cases. Case C (simulation) has the highest average temperature (318.10 K) and the highest deviation (6.39%), while Case B has the lowest deviation (0.21%) with an average temperature close to the standard. Overall, case B is the best because has the lowest deviation and its average temperature close to the standard.

These simulations tend to show slightly higher deviations compared to journal data, indicating a potential discrepancy in temperature predictions between actual recorded data and simulated models. The CFD simulation visualization further emphasizes the intricate airflow patterns within the room, highlighting areas of high velocity and temperature that could influence the thermal comfort and efficiency of the space. It can be concluded that simulations provide valuable insights and a more consistent temperature distribution compare to the previous data from literature. More comparison with other cases can be make for more accurate judgement regarding the temperature inside the unoccupied office building.

Acknowledgement

This research was supported by Universiti Tun Hussein Onn Malaysia (UTHM) through Multi-Disciplinary Research Grant (MDR) (vot Q689). Authors also want to thank to the Faculty of Mechanical Technology and Engineering, Universiti Teknikal Malaysia Melaka and the Institute for Regenerative Energy Technology (in.RET), Nordhausen University of Applied Sciences for supporting data and technical advice.

References

- [1] Rincón Casado, A., Hajdukiewicz, M., Sánchez de la Flor, F., & Rodríguez Jara, E. (2020). Calibration Methodology for CFD Models of Rooms and Buildings with Mechanical Ventilation from Experimental Results.
- [2] Peng Wang*, Mengda Liu, Qikun WangMettam. (2024). Simulation of office air conditioning air supply based on COMSOL.
- [3] Wang, L.; Li, G.; Gao, J.; Fang, X.; Wang, C.; Xiong, C. Case Study: Impacts of Air-Conditioner Air Supply Strategy on Thermal Environment and Energy Consumption in Offices Using BES–CFD Co-Simulation. *Sensors* 2023, 23, 5958. <https://doi.org/10.3390/s23135958>
- [4] Mahrukh, M., Ashraf, B., Saleem, U., & Allauddin, U. (2021). Computational fluid dynamics analysis for ventilation and air-conditioning of an office room. NED University of Engineering & Technology.
- [5] Zhang, X., Wang, Y., & Liu, J. (2017). Air distribution optimization in classrooms using CFD. *Indoor and Built Environment*, 26(6), 781-790.
- [6] Liu, Y., Zhang, T., & Wang, J. (2019). CFD analysis of HVAC performance in laboratory settings. *Journal of Building Performance Simulation*, 12(3), 321-334.
- [7] Kim, S., Lee, J., & Park, H. (2018). Performance analysis of HVAC systems in office buildings using CFD simulations. *Building and Environment*, 144, 502-510.
- [8] Awad, M., Salah, A., & Ahmed, H. (2020). Energy efficiency improvements in HVAC systems using CFD simulations. *Energy and Buildings*, 200, 50-60.
- [9] G. Mustafa, S. B. Riffat, and M. Gillott, "CFD analysis of natural ventilation in an office building," *Energy Build.*, vol. 41, no. 12, pp. 1325-1335, Dec. 2009
- [10] S. Hassid et al., "The effect of the Athens heat island on air conditioning load," *Energy Build.*, vol. 32, no. 2, pp. 131-141, Oct. 2000.
- [11] Q. Chen, "Comparison of different k-ε models for indoor air flow computations," *Numer. Heat Transf. B*, vol. 28, no. 3, pp. 353-369, 1995.
- [12] D. Jacob and A. Schälin, "CFD analysis of ventilation performance in an office room with different heat sources," *Build. Environ.*, vol. 42, no. 12, pp. 2348-2358, Dec. 2007.
- [13] Bournet, P. E., & Boulard, T. (2010). "Effect of ventilation configuration on the distributed climate of greenhouses: A review of experimental and CFD studies." *Computers and Electronics in Agriculture*, 74(2), 195-217.
- [14] Niu, J. L., & Zhang, L. Z. (2001). "Heat transfer characteristics of ventilated rooms with cooled ceilings." *Building and Environment*, 36(9), 897-906.
- [15] N. M. S. Hassan, M. M. K. Khan, and M. G. Rasul, "Temperature monitoring and CFD Analysis of Data Centre," *Procedia Eng.*, vol. 56, pp. 551–559, 2013.
- [16] P. Array and C. T. L. Spine, "Service manual," Language Policy. 2004
- [17] W. F. S. and J. W. Jones, REFRIGERATION AND AIR CONDITIONING, 2nd ed., vol. 90, no. 4. McGraw-Hill, 1979.
- [18] J. H. Ferziger and M. Peric, *Computational Methods for Fluid Dynamics*, 3rd ed. California: Springer, 2002.
- [19] Mohajernia, B. (2023). Residual stress in metal additive manufacturing of thin-walled components: investigation and development of prediction models with respect to path planning.
- [20] Kuczyński, T., & Staszczuk, A. (2020). Experimental study of the influence of thermal mass on thermal comfort and cooling energy demand in residential buildings. *Energy*, 195. <https://doi.org/10.1016/j.energy.2020.116984>