Journal of Complex Flow, Vol. 6 No. 1 (2024) 29-37

FAZ Publishing

JCF

Journal of Complex Flow

Journal homepage: www.fazpublishing.com/jcf

e-ISSN: 2672-7374

Observation on Air Flow Distribution in Room by HVAC System using Ansys Fluent CFD Simulation

Alvin Anak Boniface Barau[1], Ahmad Najmuddin Zambri[1], Abdirahman Ahmed Osman[1], Abdullah Mohammed Ahmed[1], Ishkrizat Taib[1], Bukhari Manshoor[1,*]

[1] Faculty of Mechanical and Manufacturing Engineering
Universiti Tun Hussein Onn Malaysia, Parit Raja, Batu Pahat, Johor, MALAYSIA

*Corresponding Author

Received 20 Feb 2024;
Accepted 23 Mar 2024;
Available online 15 April 2024

Abstract: Heating, Ventilation, and Air Conditioning (HVAC) systems are integral to maintaining a comfortable and healthy indoor environment. They regulate temperature, airflow, and air quality, ensuring optimal living and working conditions. In this study, Ansys Fluent was employed to design and optimize a single-room HVAC system, with a focus on achieving optimal thermal comfort, energy efficiency, and air quality. The research involved detailed CFD simulations to analyse airflow, thermal comfort, and velocity. Three cases will be focus for simulation work which are based on the location of the air-conditional unit in the room. The results revealed a clear relationship between air velocity and cooling effectiveness. Case 1, the baseline scenario, showed that higher velocities lead to increased cooling capabilities, as indicated by higher maximum pressures and lower average temperatures. Case 2 followed this trend but exhibited a slightly reduced cooling effect. However, it was Case 3 that stood out, presenting an optimized condition with significantly higher velocities and pressures, leading to the most substantial temperature reduction, reaching a low of 291.10 Kelvin (18.1°C) at 3.5 m/s. This optimal scenario demonstrated the importance of strategic air unit placement, resulting in enhanced efficiency, better air distribution, and optimized room airflow dynamics.

Keywords: Computational Fluid Dynamics, ANSYS, SolidWorks, Simulation, HVAC

1. Introduction

HVAC (heating, ventilation, and air conditioning) systems are essential to preserving a cosy and healthful interior atmosphere. To guarantee the best possible living and working circumstances, they control temperature, ventilation, and air quality. In order to achieve the best possible thermal comfort, energy efficiency, and air quality, a single-room HVAC system was designed and optimised in this study using Ansys Fluent. In-depth CFD models were used in the study to assess velocity, thermal comfort, and airflow. The simulation work will centre on three situations, each of which is depending on where the air conditioner is located in the space. The findings demonstrated a direct correlation between air velocity and cooling efficiency.

Conventional air conditioning systems often fall short in addressing inefficient airflow and unequal temperature distribution in office environments. This problem is especially noticeable in room when the main cooling source is a single air conditioner. The trick is to maximise the cooling effect while using the least amount of energy possible and guard against any negative consequences that could arise from different airflow patterns, including erosion or corrosion.

This study's main focus is on the function of HVAC (heating, ventilation, and air conditioning) systems. These systems function by modifying air flow and are crucial for maintaining interior temperature and air quality balance [1,2]. To provide adequate indoor air quality, the amount of air required is calculated based on the loads related to heating, cooling, and ventilation. The unit of measurement for air volumes is cubic feet per minute, or cfm [3]. Pipe/duct sizing tools are system design tools that take flow distribution and liquid/air distribution system sizing into account [4].

Most sizing tools are based on established methodologies and algorithms defined by organisations like the American Society of Heating, Refrigeration, and Air-conditioning Engineers (ASHRAE) [5], while many are proprietary software products offered or sold by

*Corresponding author: bukhari@uthm.edu.my

equipment manufacturers. Another crucial element is the way office buildings are constructed. The structural makeup of room has changed in tandem with improvements in building materials, methods, and environmental objectives. To satisfy the needs of their occupants, modern office buildings incorporate steel, curtain walls, strong concrete, and other elements. [6].

This study will contribute significantly to the field of engineering by offering a comprehensive approach to optimizing HVAC systems. It highlighted the potential for energy savings and enhanced thermal comfort through meticulous design and simulation. The findings provided valuable insights for engineers and architects in designing more efficient, comfortable, and sustainable living and working.

2. Previous Work

In the realm of Heating, Ventilation, and Air Conditioning (HVAC) systems, significant strides have been made to enhance indoor comfort and energy efficiency. Key to these advancements is the use of ANSYS Fluent, a Computational Fluid Dynamics (CFD) software, which has become instrumental in simulating fluid flow, heat transfer, and associated phenomena in HVAC systems. Interior environments depend on velocity fields and temperature distribution, studying them can be challenging. HVAC systems are utilized in enclosed spaces to give their occupants a comfortable environment. Convection is a fundamental phenomenon that regulates heat movement in enclosed spaces and maintains a pleasant temperature. While forced convection forces air over a hot item to remove heat and lower its temperature overall, natural or free convection causes hot air to rise and depart from the hotter surface.

Popovici's [7] study delved into the impact of air flow velocity on room temperature under different environmental conditions, providing insights into HVAC system functionality during various seasons. Popovici and Hudişteanu [8] extended this examination to a sociocultural building, emphasizing the crucial role of HVAC systems in maintaining optimal indoor conditions, particularly in preserving artwork integrity through humidity control.

Patel and Dhakar's [9] research focused on the distribution of air cooling and temperature in a room, revealing how the location of air conditioner ducts can significantly influence room temperature and achieve uniform cooling. Complementing this, Kumar and Bartaria [10] conducted an experimental analysis on air conditioning efficiency, demonstrating that rooms with a double AC duct system could cool more effectively. Kang et al. [11] explored displacement ventilation and radiant cooling, offering a detailed understanding of these systems' operational principles and their impact on thermal comfort.

Sarma et al. [12] used CFD to investigate the relationship between air conditioner unit placement and temperature distribution within a room, providing a clear correlation between these factors. Experimental and numerical research on temperature distribution and airflow in air conditioning systems under varying human loads. Their findings underscore the importance of strategically placing exhaust ports and air inlet vents to improve thermal comfort and airflow characteristics. [13]. Another study [14] examined the effect of a wood fleece insulation layer on indoor thermal comfort, highlighting its potential in enhancing the projected mean vote (PMV) for warm comfort and energy efficiency, while Vladut et al. [15] presented a case study at a medical facility, focusing on the influence of a portable air conditioning system on existing airflow patterns.

Various data center configurations was explore [16] to determine the most effective cooling strategies, providing insights into thermal mapping and the performance of computer room air-conditioners (CRACs). Their study offered a detailed 3D analysis of cool air growth through server farms, identifying high-temperature zones and optimizing cooling techniques. Further, Fulpagare et. al. [17] assessed room air currents in relation to temperature and speed, alongside thermal comfort. Their research highlighted how the placement of climate control systems affects room airflow, with implications for energy conservation and thermal comfort.

Study on the human body's heat exchange coefficient in various cooling scenarios, such as convective air heating and radiant floor heating was performed by [18,19]. Their findings indicated that radiant floor warming is more energy-efficient than convective air heating, providing valuable insights for residential building designs. Taher et al. [20] investigated the impact of ventilation and cooling supply locations on airflow properties in a lecture room. Their work emphasized the effectiveness of airflow systems in removing sensible and latent loads, thereby enhancing indoor air quality and achieving thermal comfort.

From these previous works, it can be concluded that encapsulates a diverse range of studies, each contributing to a deeper understanding of HVAC system dynamics. The findings collectively demonstrate the importance of strategic system design and placement for optimizing indoor air quality, thermal comfort, and energy efficiency. The use of CFD tools, particularly ANSYS Fluent, emerges as integral to these studies, facilitating detailed analysis and aiding in the development of more effective HVAC solutions. This body of research not only provides insights into current practices but also lays the groundwork for future innovations in HVAC system design and implementation.

3. Methodology

The primary objective was to conduct a comprehensive CFD simulation to analyse the performance of a one air conditioner within an HVAC system. Specific focus areas included the assessment of airflow patterns, temperature distribution, velocity distribution, and pressure distribution. The simulation aimed to provide insights into the design and operational effectiveness of the air conditioner in the HVAC system.

3.1 Geometry of Case Study

The geometric model of the HVAC system for a room, along with the surrounding air domain, was developed in solidwork software and imported into ANSYS Fluent. Simplifications to the geometry were made to reduce computational requirements while retaining essential features for accurate simulation. This balance ensures fidelity in capturing the crucial aspects of the fan heater's functionality.

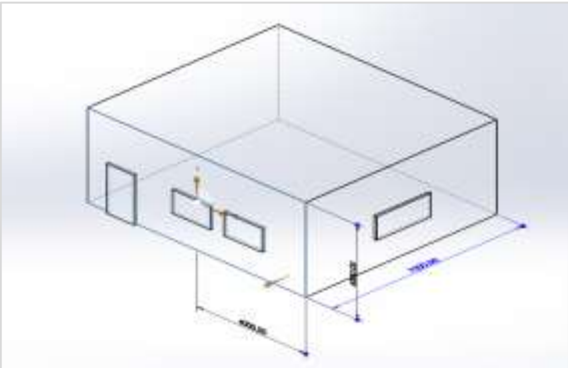


Fig. 1 - Example of the model room by using solidworks software.

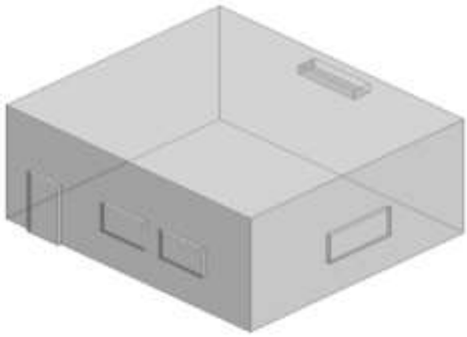


Fig. 2 - Case 1: Air-conditioner model at the center along the z-axis on the wall ANSYS.

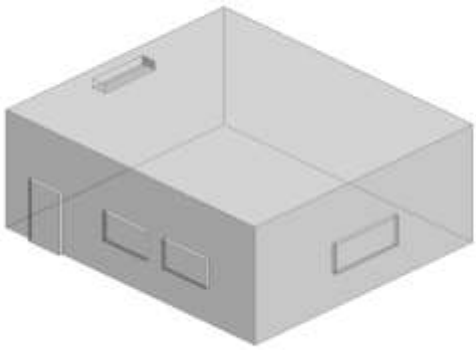


Fig. 3 - Case 2: Air-conditioner model at the center along the x-axis on the wall ANSYS.

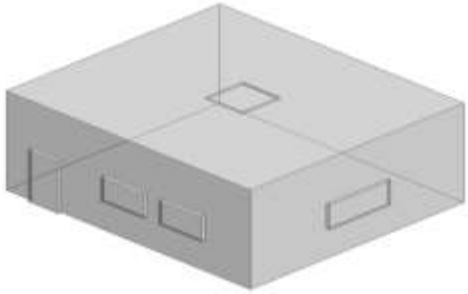


Fig. 4 - Case 3: Air-conditioner model at the center on the ceiling

The subject of this research is a residential room centrally located in a building in Parit Raja Kampus Bandar, Johor, Malaysia. This room, chosen for its representativeness in cooling analysis, measures 8m x 7m x 4m (LxWxH). The study assumes a summer outdoor ambient temperature of 313 Kelvin (40 °C). For cooling comfort, the target indoor air temperature is set at 291 Kelvin (18°C), in alignment with standard comfort ranges. The position of the indoor unit for easc case summarises in Table 1. Detail location was illustrated in the figure 1 – 3.

Table 1 – Dimension parameter

Parameters	Position
Case 1	Centre at z position on the wall
Case 2	Centre at x position on the wall
Case 3	Centre position on the ceiling of the room

3.2 Governing Equation

The Navier-Stokes Equations describe fluid motion by accounting for changes in velocity over time and space, considering factors like fluid density, velocity, pressure, viscous stresses, and external forces. They can be expressed as,

$$\rho\left(\frac{\partial t}{\partial u}+u.\nabla u\right)=-\nabla p+\nabla T+f \qquad (1)$$

where ρ is the fluid density, u is the velocity vector, t is time, p is pressure, T is the stress tensor (including viscous stresses), and f represents external forces.

In contrast, the Continuity Equation, especially for incompressible flows, ensures mass conservation by stating that the divergence of the velocity field is zero. Lastly, the Energy Equation is crucial in scenarios involving heat transfer, such as air conditioning systems where it calculates temperature distribution within the fluid by considering specific heat, temperature, thermal conductivity, and internal heat generation. It can be expressed as,

$$\rho c_p\left(\frac{\partial t}{\partial T}+u.\nabla T\right)=\nabla\left(k\nabla T\right)+Q \qquad (2)$$

Published by FAZ Publishing
http://www.fazpublishing.com/jcf

3.3 Mesh Generation and Model Solver Set up

A computational mesh was generated over the geometry. Special attention was given to areas of high-gradient potential, such as near the fan blades and heating elements, where a finer mesh was employed. This approach ensures that the mesh adequately captures critical details necessary for an accurate simulation. Thus, mesh encryption is performed at the air-conditioning inlet and outlet, and the grid independence is verified for each group of simulations to ensure independence of the number of wires. Additionally, mesh skewness was set to 0.8 where to measure of the extent to which a cell (or element) deviated from its ideal shape. For most simulation types in Fluent, the ideal shape was typically an equilateral triangle (in 2D) or a regular tetrahedron (in 3D). The figure below depicts the steps taken during the simulation of an air-conditioning system in a room, focusing on varying positions or locations of the air-conditioning unit.

3.4 Discretization and Boundary Condition

Boundary conditions were set to reflect realistic operational scenarios of the air conditioner in the HVAC system. This included specifying inlet and outlet conditions for airflow, wall conditions, and heat sources. The simulation was initialized with an estimated solution to facilitate convergence. For instance, the walls, floor, and ceiling were set as solid walls with specific thermal properties. The air conditioning unit was modelled as an inlet with a specified temperature and flow rate.

Table 2 – Summary of solver setting

Features	Setting
Flow	3D-incompressible
Formulation	Absolute
Solver type	Pressure-based
Time	Steady
Energy Equation	Enable
Solution method	SIMPLE
Material	Brick Wall
Momentum	Second Order Upwind
Velocity inlet	1.5 m/s
Inside Temp.	18°C
Outside Temp	40°C
Wall condition	Stationery and no slip condition

Table 3 – Parameter for different cases in term of velocity

Parameters	Inlet velocity		
Case 1	1.5 m/s	2.5 m/s	3.5 m/s
Case 2	1.5 m/s	2.5 m/s	3.5 m/s
Case 3	1.5 m/s	2.5 m/s	3.5 m/s

3.5 Grid Independence Test

A grid independence test in Computational Fluid Dynamics (CFD) was a systematic approach used to ensure that the simulation results were not excessively influenced by the size or refinement of the computational grid. This procedure began with an initial mesh, followed by a simulation to record key outcomes, and then proceeded with systematic mesh refinement and repeated simulations. After each iteration, key results such as pressure drop, temperature, or velocity profiles were compared to detect significant differences. In this study, velocity profile was selected to perform the GIT. The optimal mesh was selected as the one where further refinement led to negligible changes in results, achieving a balance between computational efficiency and accuracy. Thorough documentation of this process, often including a convergence plot, was essential for verifying the reliability of the CFD simulation results. Thus, in this GIT test, case 3 was chosen to perform the test. Table 4 - 5 below show the result obtained.

Table 4 – Grid independent test

Running no.	Nodes	Elements	Target skewness applied	Velocity	Distance
R-1	20908	143227	0.70	0.26383	4
R-2	27987	146737	0.76	0.31090	4
R-3	28007	146855	0.80	0.31183	4

Table 5 – Velocity at different height

R-1		R-2		R-3	
Distance (m)	Max. velocity (m/s)	Distance (m)	Max. velocity (m/s)	Distance (m)	Max. velocity (m/s)
0	1.7014	0	1.6557	0	1.7232
0.8	1.5956	0.8	1.5980	0.8	1.5910
1.6	1.5923	1.6	1.5877	1.6	1.5968
2.4	1.5727	2.4	1.5537	2.4	1.5648
3.2	1.4246	3.2	1.4276	3.2	1.4950
4	1.4403	4	1.4519	4	1.4414
0	1.7014	0	1.6557	0	1.7232

In the executed Grid Independence Test (GIT), the analysis revealed that the trend line between the R-2 and R-3 exhibited a notable convergence, with minimal deviations in velocity profiles despite further grid refinement. This observation was corroborated by the graphical representation, where an increase in the number of nodes and elements corresponded to a convergence in velocity profiles, suggesting an approach towards grid independence.

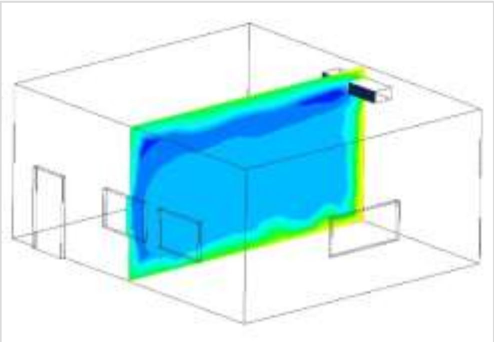
Additionally, the marginal differences observed in the velocity profile curves for runs R-1, R-2, and R-3 indicated that subsequent modifications in the mesh were exerting a progressively reduced impact on the velocity field, aligning with the primary objectives of a GIT. Furthermore, as delineated in the accompanying table, the percentage variation in velocity between runs R-2 and R-3 was recorded at 0.726%, a value substantially below the threshold of 5%. This outcome denotes the attainment of the desired accuracy. Consequently, the grid resolution established in run R-3 was chosen as the appropriate choice for subsequent simulations and it was sufficient for an accurate simulation.

4. Results and Discussion

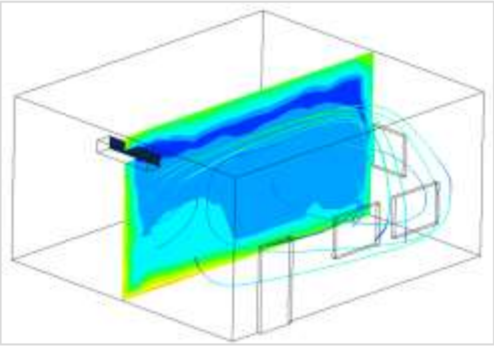
4.1 Cases and velocity to determine the three parameters

Table 3 illustrated the cases and it average velocity. However, in this study explained three cases by considering parameters included (velocity, pressure, and temperature). For the room temperature chosen 313 kelvin which is higher than the comfort temperature due to the flow which is 291 kelvins. On the other hand, for the case1, case 2 and case3 were set average velocity of 1.5,2.5 and 3.5 m/s, respectively. Moreover, the wall temperature of the room with the effect environment temperature is used to reach is 313 kelvins.

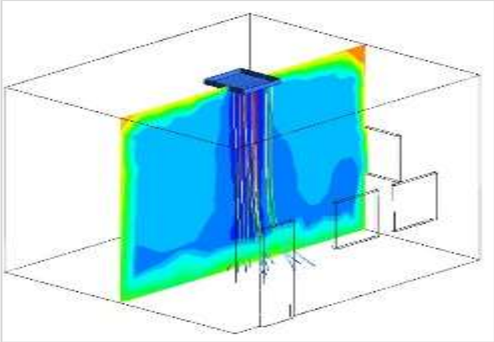
Figure 5 to 7 below depicts the results obtained for case 1, case 2, and case 3, all at a velocity of 1.5 m/s. It includes the velocity, pressure, and temperature contours obtained from these cases.



Case 1

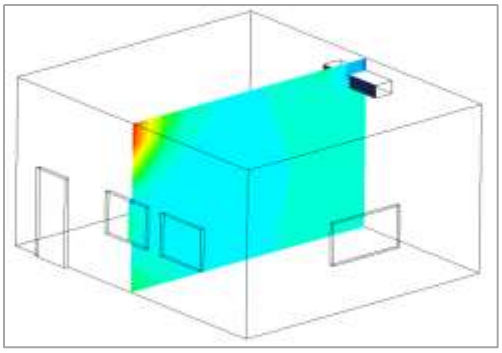


Case 2

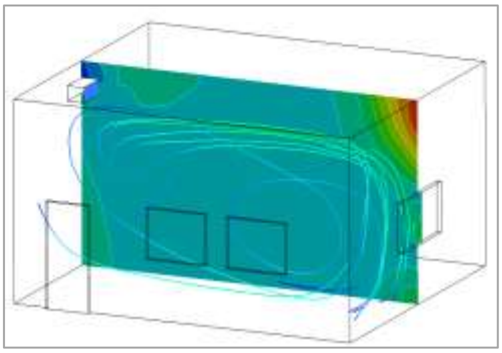


Case 3

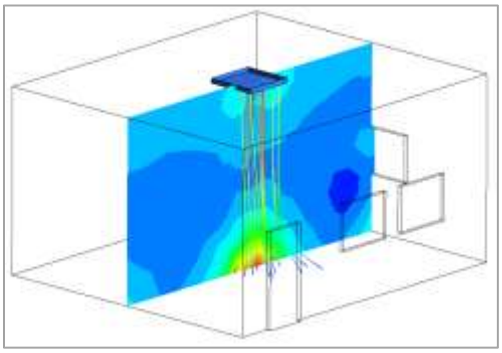
Fig. 5 - Temperature contour result obtained in ANSYS.



Case 1



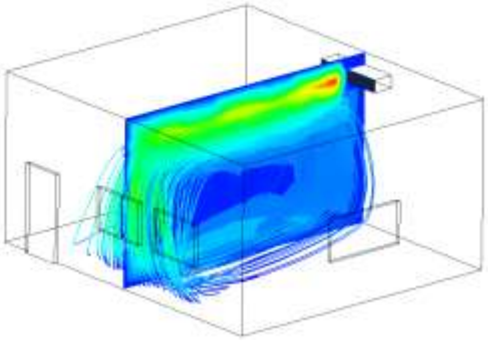
Case 2



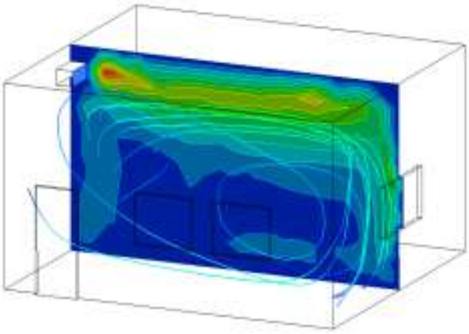
Case 3

Fig. 6 - Pressure generated due to when air flow of AC duct.

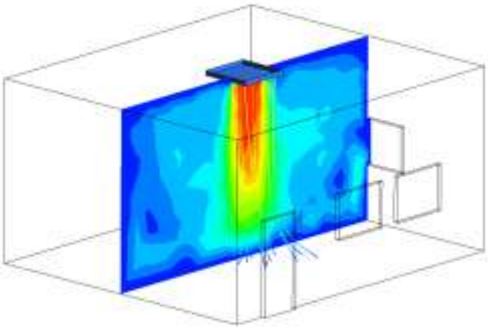
Published by FAZ Publishing
http://www.fazpublishing.com/jcf



Case 1



Case 2



Case 3

Fig. 7 - Velocity contour and flow generated due to when air flow of AC duct.

4.2 Comparison temperature, pressure and velocity for each case

In the analysis of air conditioning performance across three distinct configurations as simulated in ANSYS Fluent, the data revealed the relationship between air velocity and the resultant cooling effects within a room. Case 1 established a baseline, showing increased cooling capability with higher velocities, evidenced by the rise in maximum pressure and a decrease in average temperature, as shown Table 6. Case 2 maintained this trend but with a slightly less pronounced cooling effect, indicated by the higher average temperatures observed across similar velocities, where the lowest value obtained in case 2 was 20.14 °C at 3.5 m/s, particularly when compared to Case 1. Case 3 represented an optimized scenario where the average velocities and maximum pressures were significantly elevated for each velocity.

Table 6 – Different velocities for various cases

Vel. (m/s)	Avg. Vel (m/s)	Max. pressure (Pa)	Avg. Temp (K)
Case 1			
1.5	0.307374	0.399951	294.582
2.5	0.562674	1.14005	293.794
3.5	0.78763	3.5891	293.21
Case 2			
1.5	0.3071	0.358676	294.815
2.5	0.504327	0.989955	293.866
3.5	0.719862	1.980940	293.314
Case 3			
1.5	0.471725	0.735272	292.717
2.5	0.748533	2.12891	292.386
3.5	1.06218	4.2991	291.91

In addition, the pattern showed that the strategic placement of air units in Case 3 maximized the air conditioner's efficiency, providing a more effective cooling solution as reflected by the higher velocity outputs and lower temperature readings. This trend was consistent with enhanced air distribution and optimized room airflow dynamics. In consideration of the three scenarios differing in air conditioning parameters, Case 3, characterized by an air velocity of 3.5 m/s, is deemed most suitable for human occupancy. Figure 8 - 10 presents the data gathered from three distinct cases, providing empirical support for the findings. The findings from these simulations were significant for the design and implementation of air conditioning systems, highlighting the importance of proper air unit placement to achieve desired thermal comfort levels efficiently.

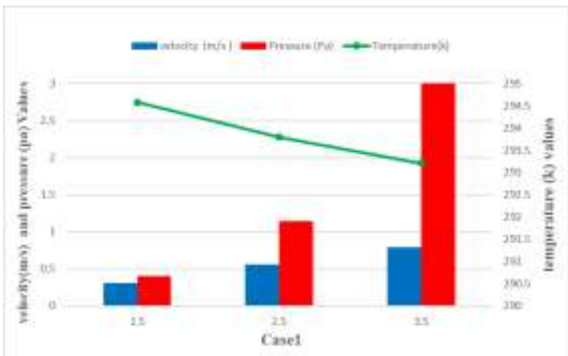


Fig. 8 - Average velocity, max. pressure and average temperature for case 1

Published by FAZ Publishing
http://www.fazpublishing.com/jcf

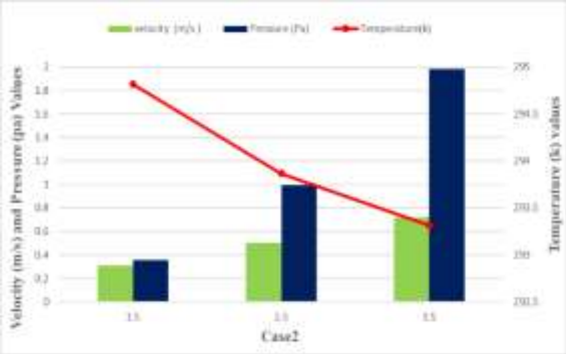


Fig. 9 - Average velocity, max. pressure and average temperature for case 2

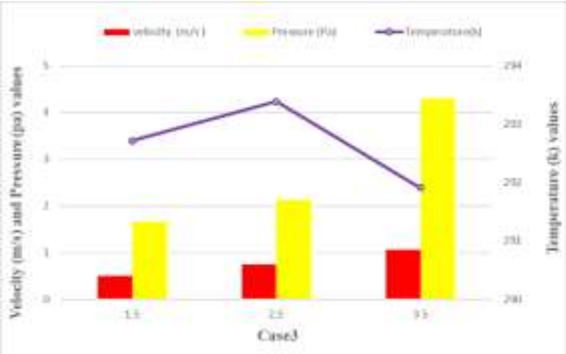


Fig. 10 - Average velocity, max. pressure and average temperature for case 3

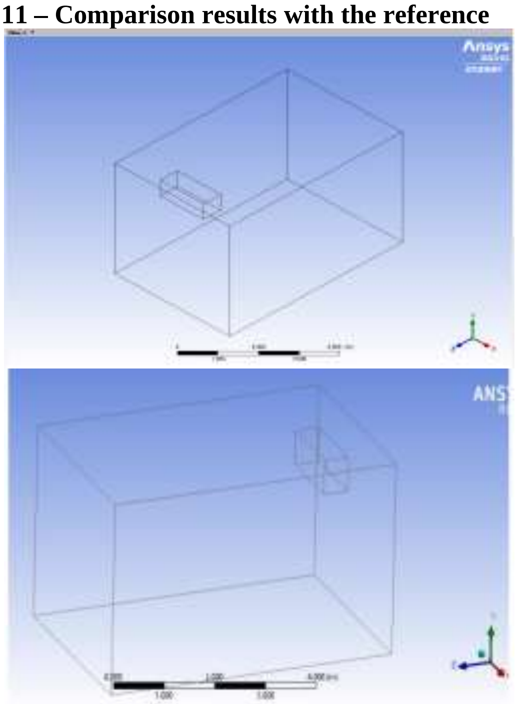
4.3 Data verification

Figure 11 shows the room design used to verify the data, which was the same as the one referenced in the paper that formed the basis for this case study.

Fig. 11 – Comparison results with the reference

Present model

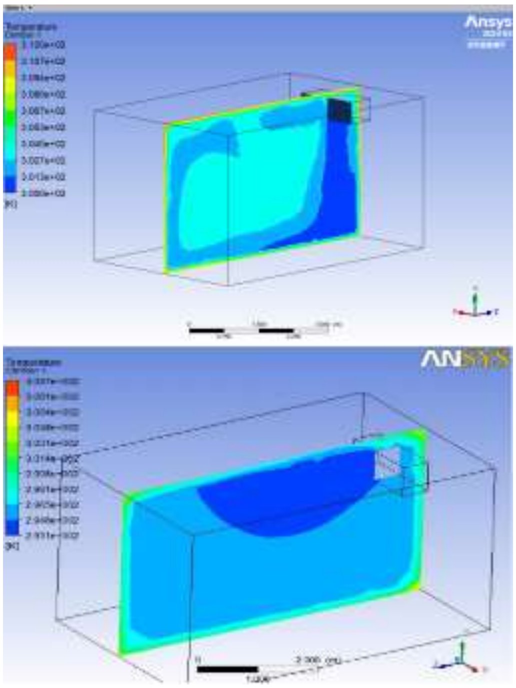
Patel & Dhakar model



The design by Patel & Dhakar [9] and present design

Present model

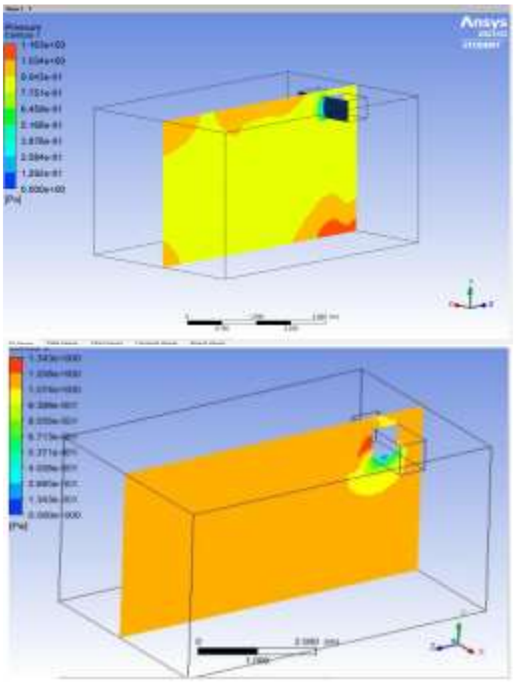
Patel & Dhakar model



Comparison of temperature

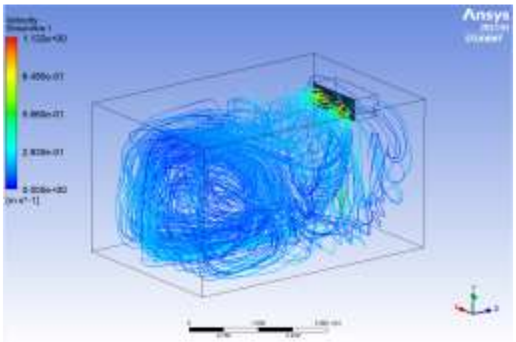
Present model

Patel & Dhakar model

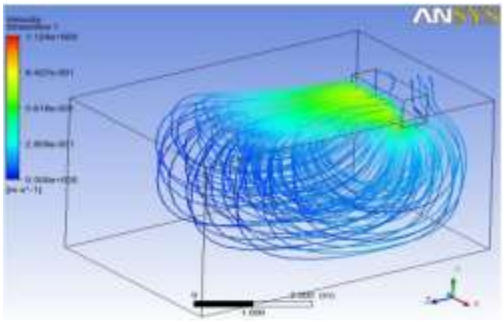


Comparison of pressure

Present model



Patel & Dhakar model



Comparison of streamlines

The subsequent calculation delineates the percentage difference between the results of the current experiment, which utilised the same model as the preceding thesis study by Patel and Dhakar [9]. Based on the calculation of percentage differences, it can be stated that the results obtained were nearly identical to the data from the previous study conducted by Patel and Dhakar which served as a reference. This close alignment provides a strong justification for the accuracy of the current case study data. The consistency in results validates the outcomes of the simulations conducted for case 1, case 2, and case 3, which involved the analysis of an air-conditioner within an HVAC system in a room, modelled in ANSYS Fluent. Thus, the similarity in findings reinforces the reliability of the simulation methods and parameters used in the current study, confirming that the simulated data accurately reflects the behaviour of the HVAC system in various scenarios.

5. Conclusion

The study explored the operation of a room equipped with one HVAC system using a comprehensive Computational Fluid Dynamics (CFD) simulation in ANSYS Fluent. The primary objective of the simulation was to shed light on the functionality and design of air conditioners within Heating, Ventilation, and Air Conditioning (HVAC) systems. Specific areas of focus included the evaluation of airflow patterns, temperature distribution, velocity distribution, and the efficiency of pressure distribution. According to the simulation findings, the HVAC system was successful in maintaining the required temperature, airflow, and pressure within reasonable limits. The results revealed that the temperature distribution within the room remained consistent, and the airflow patterns conformed to the designed parameters. Furthermore, it was observed that the pressure distribution stayed within permissible limits, indicating the system's overall effectiveness.

Overall, the simulation results indicated that the HVAC system was capable of delivering optimal thermal comfort, while also considering the energy efficiency and air quality of the space. The study provides valuable insights into the design and optimization of HVAC systems through CFD modelling, which can be utilized to enhance the efficiency and performance of HVAC systems in various environments.

Acknowledgement

The authors would like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for its support in providing the software for this study.

References

[1] Seyam, S. (2018). Types of HVAC systems. HVAC System, 49-66

[2] Bitler, T. (2023, August 16). What Is HVAC? U.S. News & World Report 360 Reviews.

[3] Sugarman, S. C. (2020). HVAC fundamentals. Crc Press.

[4] Trčka, M., & Hensen, J. L. (2010). Overview of HVAC system simulation. Automation in construction, 19(2), 93-99.

[5] WBDG, Energy Analysis Tools, Whole Building Design Guide, 2009, Available at: http://wbdg.org/ [Accessed: August, 2009]

[6] D. Amaripadath and S. Attia, “Performance dataset on a nearly zero-energy office building in temperate oceanic climate based on field measurements,” Data Br., vol. 48, 2023, doi: 10.1016/j.dib.2023.109217.

[7] Popovici, C. G. (2017, March 1). HVAC System Functionality Simulation Using ANSYS-Fluent. Energy Procedia.

[8] Popovici, C. G., & Hudişteanu, V. S. (2016, January 1). Numerical Simulation of HVAC System Functionality in a Sociocultural Building. Procedia Technology. https://doi.org/10.1016/j.protcy.2016.01.113

[9] Patel, A., & Dhakar, P. S. (2018, December 24). CFD Analysis of Air Conditioning in Room Using Ansys Fluent. ResearchGate. https://www.researchgate.net/publication/336603909_CFD_Analysis_of_Air_Conditioning_in_Room_Using_Ansys_Fluent

[10] Kumar, A., & Bartaria, V. N. (n.d.). CFD analysis of room with air conditioner by using ANSYS Workbench. JETIR. Retrieved January 7, 2024, from https://www.jetir.org/view?paper=JETIR1807263

[11] Kang, Peng, & Chenga. (2017). Analysis of Condensation and Thermal Comfort of Two Kinds of Compound Radiant Cooling Air Conditioning Systems Based on Displacement Ventilation. 10th International Symposium on Heating, Ventilation and Air Conditioning, ISHVAC2017, October 2017

[12] Sarma, S., & Jakhar, O. P. (2016). Computational Analysis of Impact of The Air-Conditioner Location On Temperature And Velocity Distribution In An Office-Room. International Research Journal of Engineering and Technology (IRJET), 3(9).

[13] Pillai, Bhand, & Shinde. (2016). A Review on CFD Analysis in Air-Conditioning System. International Journal of Current Engineering and Technology.

[14] Prakash, D. (2015). Transient analysis and improvement of indoor thermal comfort for an air-conditioned room with thermal insulations. Ain Shams Engineering Journal, Elsevier.

[15] Vladut, Sbirna, & Sebastian. (2014). CFD simulation of the airflow pattern within a three-bed hospital room with or without a portable air conditioner in use. 18th International

[16] Hassan, Khan, & Rasul. (2013). Temperature monitoring and CFD Analysis of Data Centre. Procedia Engineering, 56, 551–559. Elsevier.

[17] Fulpagare, & Agrawal. (2013). Experimental Investigation On Room Air Flow Pattern & Thermal Comfort Quantification. International Journal of Engineering Sciences & Emerging Technologies, 6(1).

[18] Lee, S., Nogami, M., & Yamaguchi, S. (2013). Evaluation of Heat Transfer Coefficients In Various Air conditioning Modes By Using Thermal Manikin. Proceedings of the 13th Conference of the International Building Performance Simulation Association.

[19] FAZ Farid Sies, M., Ismail, N. T. J., & Ramli, M. N. (2020). Airflow Analysis of Air Conditioning System for Lecture Hall: Study Case – Block G3, UTHM. *Journal of Complex Flow*, *2*(2), 1-5.

[20] Tahersima, Stoustrup, & Rasmussen. (2010). Thermal Analysis of an HVAC System with TRV Controlled Hydronic Radiator. Proceedings of the 6th Annual IEEE Conference on Automation Science and Engineering.