

Air Conditioning of an Office by Two Fans

Abobakr Saleh Yaslam Balfaqih¹, Amir Faiz Razali¹, Christophe Kordelli Juut Anak Charles Kored¹, Mohamad Hannan Asyraf Hasnul Fauzi¹, Ishkrizat Taib^{1,*}

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

*Corresponding Author

Received 12 Feb 2024;
Accepted 3 Mar 2024;
Available online 15 April 2024

Abstract: The study employs Computational Fluid Dynamics (CFD) to analyse the air conditioning of an office space using two fans. The research utilizes simulations in ANSYS Fluent to visualize and assess airflow and temperature distribution patterns. Findings indicate areas of uneven cooling and air circulation, which are critical for maintaining comfortable and energy-efficient environments. The study highlights the importance of strategic fan placement and HVAC system optimization to enhance indoor air quality and thermal comfort in office settings

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

1. Introduction

In today's fast-paced and technology-driven world, the significance of a comfortable and efficient workspace cannot be overstated. The role of air conditioning systems, particularly in maintaining optimal indoor environments, is central to achieving this goal. This executive summary presents an in-depth analysis of air conditioning for office space using two fans, leveraging the principles of Computational Fluid Dynamics (CFD).

In many office settings, conventional air conditioning systems fail to adequately address uneven temperature distribution and airflow inefficiencies. This issue becomes particularly pronounced in offices equipped with only one air condition unit as their primary cooling mechanism. The challenge lies in maximizing the cooling effect while minimizing energy usage and preventing potential adverse effects such as erosion or corrosion due to airflow patterns.

The role of HVAC (Heating, Ventilation, and Air Conditioning) systems is central to this study. These systems, essential for balancing indoor temperatures and air quality, work by adjusting air flow [1,2]. Based on the loads associated with heating, cooling, and ventilation, the amount of air needed to produce sufficient indoor air quality is calculated. Cubic feet per minute (cfm) is the unit of measurement for air volumes [3]. System design tools that consider flow distribution and liquid/air distribution system sizing are known as pipe/duct sizing tools [4].

While many sizing tools are proprietary software packages supplied or sold by equipment manufacturers, the majority are based on standard techniques and algorithms defined by organizations such as the American Society of Heating, Refrigeration, and Air-conditioning Engineers (ASHRAE) [5]. In other words, the intricate dynamics of air movement, temperature control, and energy consumption become pivotal in defining the efficacy of these systems.

Climate change also plays a significant role. In Malaysia, there have been notable changes in weather patterns, which also will manipulate the temperature as shown in figure 1 [6]. These changes affect how HVAC systems should be designed and operated.



Fig.1 - Annual moving average of mean daily temperature in Malacca (1960–2018)

The structure of office buildings is another important factor. Office buildings have evolved structurally along with advancements in materials, construction techniques, and sustainability goals. Modern office buildings integrate heavy concrete, steel, curtain walls, and other features to meet occupant needs. [7].

Finally, the use of software like SolidWorks and ANSYS is essential. These tools allow for detailed modeling and simulation of air conditioning systems, ensuring they meet the demands of modern buildings and changing climates. ANSYS is a versatile finite element analysis program used for structural and thermal calculations, offering features like heat transfer, fluid flow, elastic and plastic behavior, and dynamic analysis. It allows for one, two, or three-dimensional analysis, and includes a comprehensive suite for pre- and post-processing [8]. On the other hand, SolidWorks, developed by Dassault Systèmes, is a 3D computer-aided design (CAD) software that excels in parametric design, allowing interconnected modifications across part, assembly, and drawing files. It's user-friendly and widely used for both 2D and 3D modeling, providing an extensive range of features for creating and modifying designs. Together, these tools are indispensable in modern engineering, facilitating detailed design and analysis across various industries. [9,10].

This study aims to connect these various aspects, providing a comprehensive view of how to effectively use two fans for air conditioning in an office. It combines scientific, engineering, environmental, and technological insights to achieve comfortable and efficient indoor environments.

2. Methodology

In this section, brief methodology will be present. The process for the simulation process starting is the data collection from previous literature, design the office, meshing process, set up the boundary condition and run the simulation. The process of designing an office with an air conditioning unit and two fans was done using SolidWorks software while the data analysis was carried out by using Ansys Fluent software.

2.1 Geometry of Case Study

The office has overall square dimensions of 10 meters by 10 meters with a ceiling height of 3 meters. The geometry of ventilation system in CFD. The office space can be divided roughly into four quadrants. The northern part of the office consists of two workstations on either side with enough space for walking in between leading to the back of the office. The office also has one air conditioning unit and two fans to regulate the temperature.

SolidWorks is a 3D CAD modelling software that is used here to create a 3D model layout of the office space. This would involve modelling objects like desks, chairs, shelves, lighting etc and assembling them in the

virtual 10x10x3 meter room. ANSYS is an engineering simulation software with capabilities in fluid dynamics, structural analysis, heat transfer etc. Here, ANSYS would be used to simulate air flow, ventilation, and heat transfer within the office space modelled in SolidWorks. Having the complete 3D CAD model from SolidWorks allows importing geometry easily into ANSYS simulation tools. This facilitates advanced analysis around thermal comfort, and indoor air quality helping create a productive office environment.

2.2 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,

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

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,

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

where c_p is the specific heat at constant pressure, T is temperature, k is the thermal conductivity, and Q represents internal heat generation.

Together, these equations provide a comprehensive mathematical framework for analyzing and predicting the behavior of fluid flow and heat transfer in various engineering applications [11,12].

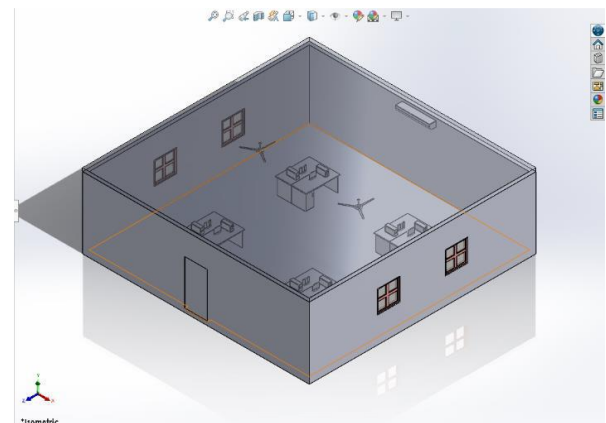


Fig.2 – 3D CAD drawing made in SolidWorks

2.3 Discretization and Boundary Condition

The type of mesh generated was tetrahedrons. Assuming steady state thermal analysis is conducted, a medium density fluid mesh containing approximately 2 million nodes and solid mesh with 500,000 nodes would sufficiently capture airflow patterns and surface heat transfer from computers, lighting, and occupants. Mesh refinement would concentrate on boundary layer regions close to heat dissipating equipment, HVAC diffusers, outlets and inlets to accurately map thermal gradients within the enclosed office space at varying peak conditions over time.

High quality mesh with maximum skewness under 0.95 would be generated as a best practice. Known problematic spots would be rigorously assessed to ensure smooth element aspect ratios as they transition from fine grain geometrical face edges to coarser outer walls and ceiling. In designing an HVAC system for an office environment, several key parameters must be considered to ensure optimal performance. The inlet conditions start with a typical air conditioning temperature set at 22°C, reflecting standard comfort levels. The velocity of air is determined by the specifications of the air conditioning unit, commonly ranging from 0.1 to 0.5 m/s, indicative of a gentle breeze. Flow rate, another critical factor, is defined based on the HVAC system's capacity, with residential systems varying from 0.1 to 1 m³/s [13].

For outlet conditions, the pressure at the outlets is typically set to atmospheric pressure, which is about 101325 Pa at sea level, adjusted as necessary for local atmospheric conditions. The flow direction is configured to allow air to exit the room naturally, with calculations often performed by simulation software.

Wall conditions involve specifying thermal properties like insulation R-value or U-value. A standard insulated wall may have an R-value between 13 to 19 or a U-value of 0.05 to 0.08 W/(m²K). Thermal conductivity for common building materials varies, with brick at 0.6 to 1.0 W/m.K, concrete at 0.8 to 2.0 W/(m.K), and insulation materials at 0.02 to 0.04 W/m.K [14]. Specific heat transfer through glass windows ranges with U-values from 5.8 for single-pane to 1.1 for high-performance double-glazing. Specific heat capacities are around 880 J/kg.K for concrete, 1200 to 2400 J/kg.K for wood, and approximately 840 J/kg.K for glass [15]. The fluid properties in HVAC systems typically involve air, with a standard density assumption for incompressible flow at 1.2041 kg/m³, a figure applicable at 20°C [16].

Thermal properties include the specific heat capacity of air (c_p) at room temperature, approximately 1005 J/kg.K, vital for calculating the energy needed to heat the air. The thermal conductivity of air at room temperature is around 0.0257 W/m.K, essential for heat transfer calculations [17]. Fan specifications for an office setting typically involve a flow rate of about 1 m³/s for ceiling fans, with velocity ranging from 1 to 5 m/s for standard office fans, and potentially higher for high-speed fans [13].

Table 1 – Boundary condition

Parameters	Values
Size of the domain	10m x 10m x 3m
Type of fluid	Air
Viscosity	1.81×10^{-5} Pa.s
Density	1.2041 kg/m ³
Fan Velocity	1 to 5m/s
HVAC Flow Rate	0.1 to 1 m ³ /s
HVAC Temperature	22°C
Room Temperature	24 to 28°C

3. Results and Discussion

3.1 Grid Independency Test

To vary the mesh element size and check the output result for each mesh can be found out from the Grid Independence test (GIT). The results were constant for element sizes of 0.3m and 0.2m, showing that no further refinement is needed for element sizes under 0.2m.

3.2 Velocity Profile

In the velocity profile shown in Figures 3 and 4, different colours are used to represent the speed of the air at different points in the office space: the blue to green gradient indicates lower velocities, while yellow to red represents higher velocities. The streamlines demonstrate how air is circulated by the fans, with some areas showing more concentrated airflow, likely closer to the fans, and others showing less, indicating potential areas of stagnant air or lower circulation.

The airflow of the simulations is more concentrated around the fans, with less velocity as the air moves away from the fans. This suggests that while the fans are effective in their immediate vicinity, their influence on air circulation decreases with distance. There could be areas in the room that are not receiving sufficient airflow, which could lead to uneven cooling or hot spots. This type of simulation is crucial for designing HVAC systems to ensure even air distribution in all parts of the room.

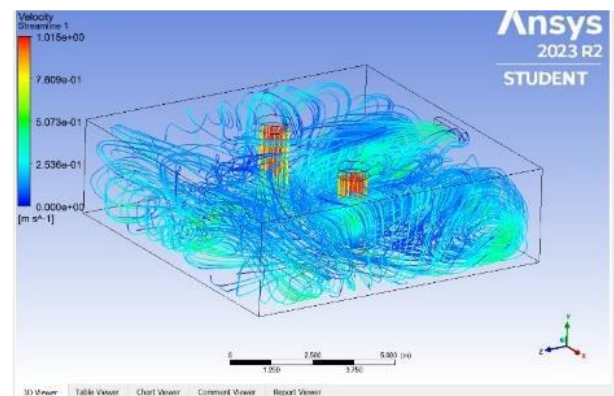


Fig. 3 - Streamline for 1 air conditioner and 2 fans

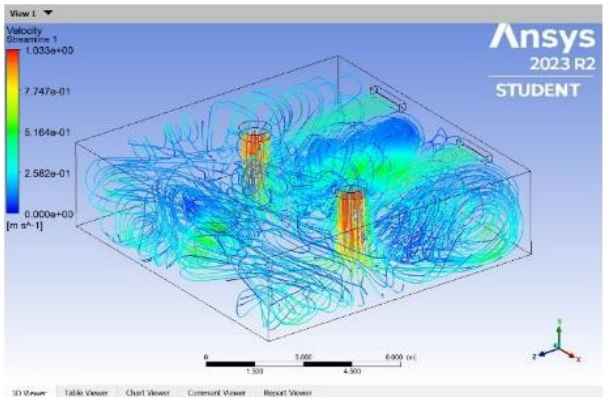


Fig. 4 - Streamline for 2 air-conditioners and 2 fans

3.3 Velocity Contour

From figures 5 and 6, it was observed that areas of high velocity (red and orange regions) and low velocity (blue regions), which suggest the effectiveness of the fan placement and settings in circulating air throughout the room. High-velocity areas are typically close to the fans or in the direct path of the airflow, while low-velocity areas might indicate regions where airflow is stagnant or not as strong.

In a practical sense, these contour plots are used to identify zones within the office that might not be receiving adequate air circulation, which is critical for ensuring uniform air distribution for comfort and efficiency. The specific values on the legend of the images allow for a detailed understanding of the airflow pattern, which can be used to adjust the placement or settings of the fans for optimal performance.

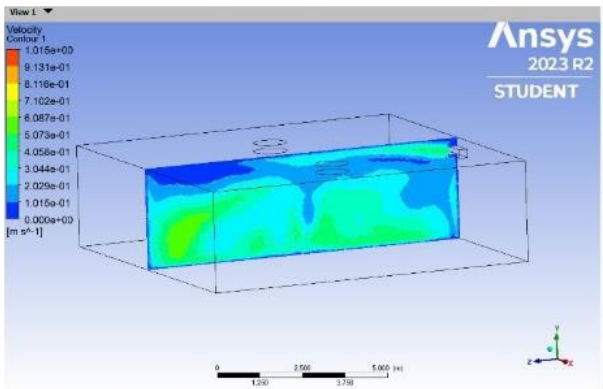


Fig. 5 - Velocity contour for 1 air-conditioner and 2 fans

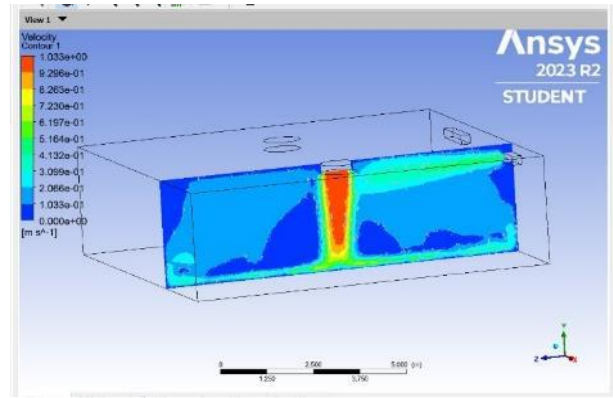


Fig. 6 Velocity contour for 2 air-conditioner and 2 fans

3.4 Temperature Contour

From Figures 7 and 8, it is observed that the range of temperatures across the space, with the warmest areas shown in red and the coolest in blue. There are some regions which are near the air conditioning outlets or the path of the cooled air, are significantly cooler than others. Conversely, the red and orange areas indicate regions that may be receiving less cooling, which could be due to various factors such as distance from cooling sources, poor air circulation, or external heat gain.

Such a simulation aims to ensure a uniform temperature distribution for comfort and energy efficiency. Adjustments to the HVAC system, including the placement of fans, vents, and the flow rates, can be made based on these results to improve the overall temperature regulation in the office space.

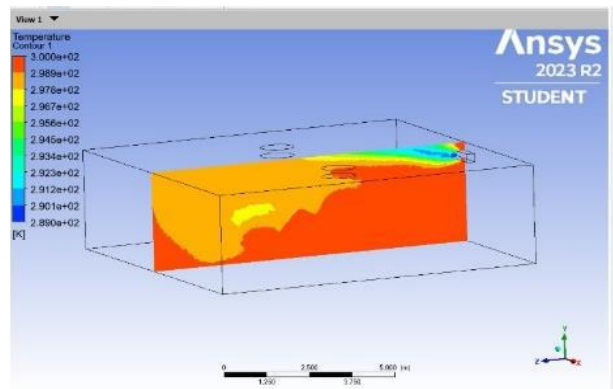


Fig. - 7 Temperature contour for 1 air conditioner and 2 fans

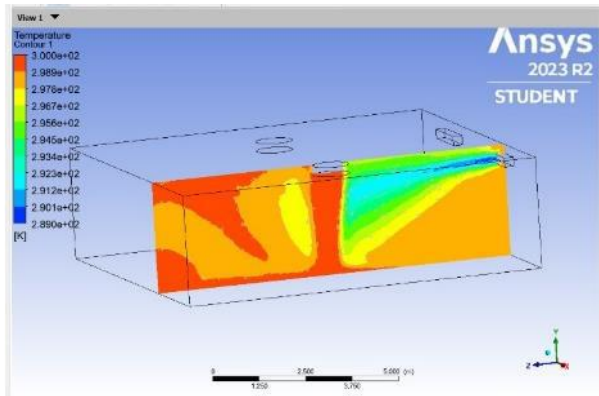


Fig. 8 - Temperature contour for 2 air conditioner and 2 fans

4. Conclusion

The study focuses on the simulation of airflow and temperature distribution in an office environment using two fans. The objective is to achieve uniform temperature and airflow, which are crucial for occupant comfort and energy efficiency. The Computational Fluid Dynamics (CFD) simulations performed with ANSYS Fluent software provided visualizations of velocity and temperature contours. From these, it was observed that while the fans effectively circulate air in their immediate vicinity, their influence diminishes with distance, leading to potential hotspots. Temperature contours revealed uneven cooling, with adjustments needed to optimize the HVAC system's performance. The study is significant for designing effective and sustainable air conditioning strategies in office spaces.

The problem goal is to investigate the effect of blown airflow within the room. Two simulations were done, one with 2 air conditioners and 2 fans while the second simulation is 1 air conditioner and 2 fans. The velocity and thermal performance of the 2 fans are easily visible from related contours and vectors. Every fan has a polynomial porous jump identified in the Fluent software. The HVAC performance of these two fans for this large office seems reasonable.

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] K. H. D. Tang, "Climate change in Malaysia: Trends, contributors, impacts, mitigation and adaptations," Sci. Total Environ., vol. 650, pp. 1858–1871, 2019, doi: 10.1016/j.scitotenv.2018.09.316.
- [7] 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.
- [8] Kohnke, P. C. (1982). Ansys. In Finite Element Systems: A Handbook (pp. 19-25). Berlin, Heidelberg: Springer Berlin Heidelberg.
- [9] Sculpteo. (n.d.). SolidWorks Definition. Sculpteo. <https://www.sculpteo.com/en/glossary/solidworks-definition/>
- [10] SOLIDWORKS. (2021). Features - SOLIDWORKS. https://help.solidworks.com/2021/english/SolidWorks/sldworks/c_features_top.htm
- [11] White, F. M. (2020). "Fluid Mechanics." 8th ed. McGraw-Hill Education.
- [12] Munson, B. R., Young, D. F., & Okiishi, T. H. (2016). "Fundamentals of Fluid Mechanics." 8th ed. Wiley.
- [13] American Society of Heating, Refrigerating and Air-Conditioning Engineers. (2020). "ASHRAE Handbook—HVAC Systems and Equipment." ASHRAE.
- [14] U.S. Department of Energy. "Building Materials Database." <https://catalog.data.gov/dataset/?tags=physics>
- [15] Engineering Toolbox. (2021). "Engineering Toolbox: Resources, Tools and Basic Information for Engineering and Design of Technical Applications." <https://www.engineeringtoolbox.com>
- [16] Çengel, Y. A., & Boles, M. A. (2020). "Thermodynamics: An Engineering Approach." 9th ed. McGraw-Hill Education.
- [17] Incropera, F. P., & DeWitt, D. P. (2019). "Fundamentals of Heat and Mass Transfer." 8th ed. Wiley.
- [18] Ashwanee Patel, P. S. Dhakar (2018) "CFD Analysis Of Air Conditioning In Room Using Ansys Fluent". <https://www.researchgate.net/publication/336459195>
- [19] Shengwei Wang, Dongsheng Zhu (2003) "Application of CFD in Retrofitting Air-conditioning Systems in Industrial Buildings". <https://www.researchgate.net/publication/237897776>
- [20] Giovanni Semprini, Giovanni Semprini, Aminhossein Jahanbin Beatrice Pulvirenti and Paolo Guidorzi (2019) "Evaluation of Thermal Comfort Inside an Office Equipped with a Fan Coil HVAC System: A CFD Approach"