Paper ID: 210

Simulation-Based Optimization of Air Conditioning Systems for Enhanced Energy Efficiency and Indoor Air Quality at Rajshahi University of Engineering and Technology

Md Jafrul Hassan[1] , Apurbo Saha[2] , Mehedi Hassan[3], Avik Biswas[4]
[1]Department of Mechanical Engineering
Rajshahi University of Engineering & Technology, Rajshahi-6204
E-mail: jafrulhasanangel@gmail.com, apurbos704@gmail.com, mehedihasan1902123@gmail.com

Abstract

This paper presents a comprehensive simulation-based optimization study of the air conditioning (A/C) system at Rajshahi University of Engineering & Technology. The great demand for energy and the growing trend toward energy-efficient HVAC systems force researchers to focus on choosing the most appropriate HVAC system for industrial buildings during the design process. The existing approaches to designing system rely on the rules of thumb due to the absence of a standardized guideline for the design process consisting of multiple ductless A/C units, which are both low efficiency and low IAQ due to the lack of fresh air intake by the ductless A/C unit. Their design product might be functional but better in performance and cost. The goal of the research is to improve the AC units' performance through the use of simulation techniques. There is 21% increase in EER & 23% increase in COP from base model.

Keywords: A/C System, Simulation, ASHRAE.

1. Introduction

Energy is crucial for human civilization and industrial activity, but its limited availability has led to increased demand for energy development. The growth of the population has increased energy extraction and resource consumption, posing challenges to resource depletion and environmental conservation. The need for energy-efficient HVAC systems in both residential and non-residential buildings is growing. However, the HVAC system at RUET is inefficient and inadequate for ventilation, leading to elevated CO_2 levels. Traditional design processes, often based on heuristic methods, are insufficient due to the lack of standardized protocols for multiple ductless A/C units. This results in low efficiency and inadequate indoor air quality, negatively impacting students and instructors, hindering effective teaching and classroom management. To address this issue, we employed ANSYS Fluent V.21 to simulate the parameter model and forecast certain scenarios, subsequently comparing them with the real settings to analyses energy efficiency, indoor air quality, and occupant comfort in accordance with ASHRAE standards.

A number of papers address the researches of CFD analysis of the air-conditioning system. Xuyang Zhong examined the effects of outdoor units' heat rejection on the building surroundings, building cooling load, and indoor PM2.5 exposure of a six-storey mixed-mode building using EnergyPlus and Fluent.[1] Based on the STTNAS classroom model, Rampravesh Yadav and Shivendra Singh investigated the distribution of air velocity, pressure, and temperature in four scenarios involving various air conditioner locations. The classroom model is numerically simulated using Fluent software with a transient pressure-based solver. The room is simulated for ten minutes with an intake velocity of 0.5 m/s and an inlet temperature of 16 ºC. Computer calculations showed that moving the air conditioner had no effect on the classroom's mean temperature. In cases 1 and 2, the measured temperature is 27.9 ºC, however in cases 3 and 4, it is 27 ºC. [2] Using ANSYS Fluent, Ashwanee Patel and P. S. Dhakar demonstrate the CFD modeling of forced ventilation on an air-conditioned room in three different duct positions and there are some significant changes in the maintained volume which are to be circulated [3]. Yosua Heru Irawan and Daru Sugati Investigates the Temperature and Air Velocity Distribution of Split-Type A/C in STTNAS classroom models and numerical simulation (turbulence model equation) is done using ANSYS Fluent 21 [4].

This project will do numerical simulations to analyze the temperature and air velocity of the air conditioning system in a classroom at Rajshahi University of Engineering & Technology. The simulations will utilize the commercial software ANSYS Fluent 21. The commercial software ANSYS Design Modeler and ANSYS Meshing will be employed for model development and meshing. The commercial software ANSYS CFD-Post will be employed to display the results.

2. Methodology

2.1.Geometry

Research is conducted on the classroom format at Rajshahi University of Engineering and Technology (RUET), Bangladesh. The dimensions of this classroom are 15.5 m in length, 5.9 m in width, and 3.2 m in height; it is equipped with three split-type air conditioning units. Numerical simulations are performed on the model of the classroom to assess the efficacy of the AC system. The A/C units are positioned in four distinct orientations as illustrated in the figure.

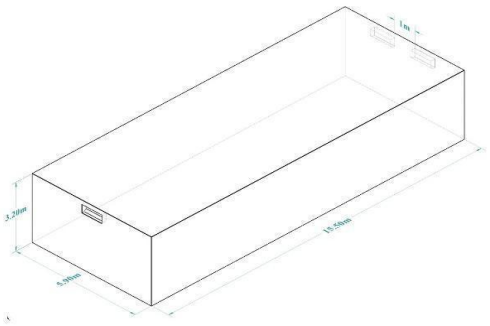


Fig. 1. Dimension of the room under investigation

- Case 1: Two positioned at the front and one at the back of the room wo in the front and one in back of the room
- Case 2: One located on left and one on right wall and one on the front wall
- Case 3: Three situated on the ceiling of the room
- Case 4: One placed on the front wall, one on the back wall, and one on the right wall

2.2.Mesh Generation and Validation

In Meshing process, the classroom domain was divided into small 3D volumes. Meshing is executed using ANSYS Meshing software. Inflation and body sizing has been employed, yielding a tetrahedral mesh. Fig. 2 illustrates comprehensive details regarding mesh production and layout. Approximately 1,500,000 mesh elements have been generated. Mesh quality evaluations are essential to avert errors throughout the iterative process, as mesh quality affects the resulting numerical solutions. The mesh quality results for this simulation reveal a skewness quality of 0.24 and an orthogonal quality of 0.75. The mesh is adequately constructed and prepared for simulation.

To ensure the accuracy and stability of the simulation results, a grid independence test was conducted. This test compared temperature and velocity distributions across various no of elements, variations in temperature and velocity distributions were minimal (<1%), establishing this mesh size as grid-independent for this study. Table 2 summarizes the findings, showing consistent values across mesh densities.

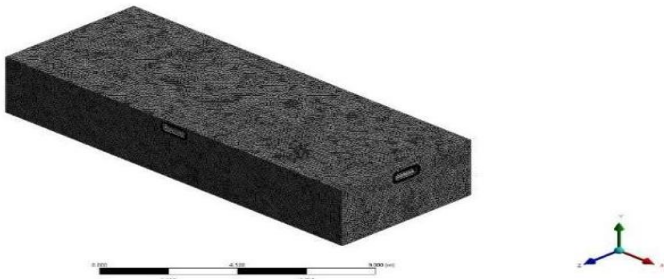


Fig. 2. Mesh generation.

Table 1. Grid Independence Test Results

Cell Count	Temperature (°C)	Velocity (m/s)	Variation
31000	298.65	2.68	<1%
100000	298.79	2.62	<1%
500000	298.83	2.65	<1%
1000000	299.04	2.68	<1%
1500000	299.05	2.68	<0.1%

Based on these results, a mesh size of approximately 1,500,000 elements was selected (Variation <0.1%).

Table 2. Different parameters on different cases

Case	Skewness	Orthogonal Quality	No. of Elements
1	0.28	0.72	1430809
2	0.24	0.75	1497404
3	0.24	0.75	1495409
4	0.27	0.73	1537637

2.3.Boundary Condition

The establishment of boundary conditions entails determining their location and orientation. This technique can be performed using ANSYS Fluent (Flow). A boundary condition consists of a velocity input, a pressure exit, and a wall. The velocity of the indoor unit was taken to be 3.2 m/s and the inlet temperature was 25°C. The room wall temperature was assumed to be 40°C. The outflow pressure is expected to be 1 atm. This boundary requirement is enforced on all walls surrounding the classroom; hence, windows and doors are also categorized as walls.

2.4.Solver Model

This is the procedure for determining solutions through iterations until a convergent approximation is attained. The Reynolds-averaged Navier-Stokes (RANS) model underpins the Reliable k-ε turbulence model employed in the present computational analysis. The primary objective of these investigations is to establish steady-state predictions for turbulent flow. Subsequently, the time-dependent flow regimes are analyzed using the LES formulation. Osborne Reynolds introduced the RANS equations, derived from the continuity and momentum equations, to analyses instantaneous values. The Realisable k-ε turbulence model is considered one of the most preferred RANS-based turbulence models. It comprises two transport equations, a novel equation for the dissipation rate, and a formulation for eddy viscosity [5]. This model was utilized in the current investigation. The reliable k-ε turbulence model attains superior performance in areas characterized by substantial flow separation and recirculation zones [6]. Moreover, the integration of a two-layered k-ε modelling technique enables the application of the model in viscous sublayers [7]. The BSL model, while less reliant on arbitrary freestream variables, aims to yield results comparable to Wilcox's original k-w model. As one approaches the boundary-layer edge, it gradually transitions to the high Reynolds number Jones-Launder k-ε model. The SST model, which is derived from the BSL model, accounts for the transfer of the primary shear stress in boundary layers with adverse pressure gradients. Evaluations of various flow fields demonstrate promising results for both models. Favorable outcomes are achieved when both models are assessed over various flow fields. The solution method employed a SIMPLE-Scheme, utilizing the conventional initialization approach from the 2018 4th International Conference on Science and Technology (ICST) in Yogyakarta, Indonesia, with an initial room temperature of 40 degrees Celsius. The space is replicated for 3 minutes, utilizing a time step size of 1 second, resulting in a total of 180-time steps. The upper limit for iterations each time step is established at 100 iterations.

2.5.Data Analysis Method

Data analysis is performed using ANSYS CFD-Post software to determine the distribution of air velocity and temperature. Two categories of observation planes were established to analyze velocity and temperature distribution. The initial plane is a horizontal plane consisting of two planes, whereas the second plane is a vertical plane including four planes. The horizontal plane consists of plane H-1, positioned 2 meters from the front wall, and plane H-2, located 4 meters from the front wall. The vertical plane consists of plane V-1 (3.1 meters from the left wall), plane V-2 (6.2 meters from the left wall), plane V-3 (9.3 meters from the left wall), and plane V-4 (12.4 meters from the left wall). The observation period is specified as the time during which the cooling system functions for 5 minutes. Figures 4 and 5 depict the defined horizontal and vertical planes.

2.6.Code Validation with Published Literatue

To validate the credibility of the simulation results, key outputs such as temperature distribution and airflow patterns were compared with similar studies in published literature:

- Comparison with Patel and Dhakar’s Study: The temperature distribution results align closely with the findings of Patel and Dhakar (2018) [1], where comparable turbulence models (Realisable k-ε) were used in air-conditioned room simulations.
- Air Velocity Analysis: The simulated air velocity patterns matched the results observed by Irawan et al. (2018) [2], demonstrating similar airflow behavior in classrooms equipped with split-type A/C units.

These comparisons provide confidence in the model's accuracy and its applicability to similar airflow and cooling performance studies, confirming that the code is consistent with established literature.

3. Results and Discussions

3.1.Temperature Distribution in the Classroom

In the standard model employed in RUET, the initial temperature was observed to be 40 degrees Celsius. Subsequently, after three minutes of the room simulation, the temperature progressively diminished over time. Enhanced cooling was seen in the V1 and d V2 plane due to the presence of two air conditioning units on the front wall. In V3, there was inadequate cooling resulting from limited air dispersion and poor mixing. The V4 aircraft was cooler due to the air conditioning on the rear wall. In the H1 and H2 plane, it is evident that the upper section of the front wall experiences reduced cooling due to the high velocity of air from two air conditioning units. Furthermore, owing to the considerable distance between the front and rear walls. After 3 minutes, the temperature was recorded at 29 degrees Celsius. The statistic indicates that this was outside the ASHRAE recommended comfort threshold.=These arrangements provide a clearer insight into the impact of AC configuration on room cooling. This study analyzed the temperature contour and velocity vector.

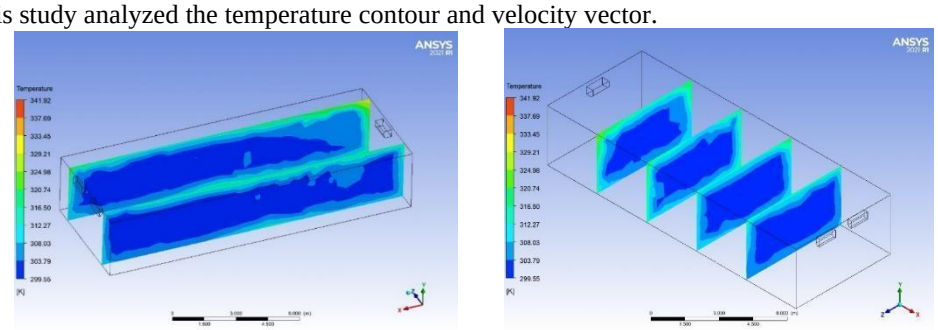


Fig. 3. Case 1 Temperature distribution in horizontal plane(left) and vertical plane(right)

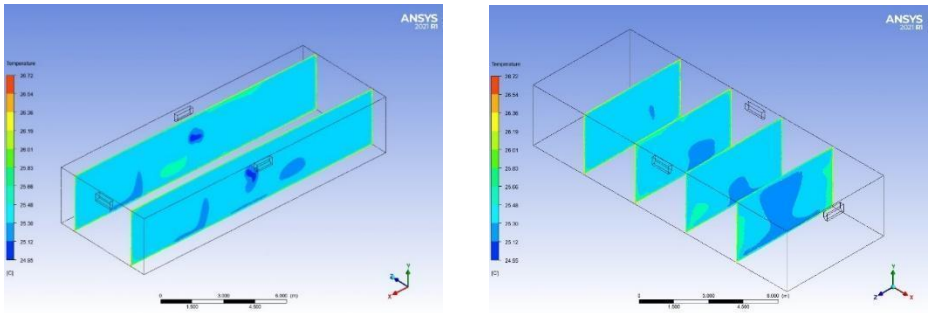


Fig. 4. Case 2 Temperature distribution in horizontal plane(left) and vertical plane(right)

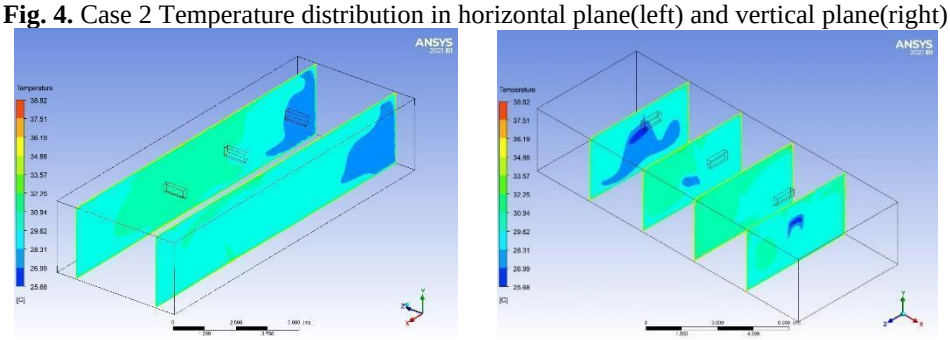


Fig. 5. Case 3 Temperature distribution in horizontal plane(left) and vertical plane(right)

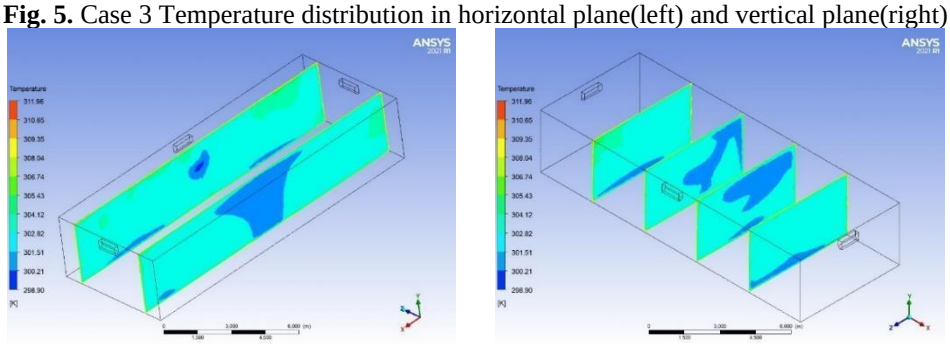


Fig. 6. Case 4 Temperature distribution in horizontal plane(left) and vertical plane(right)

In Case 2, the temperature contours depicted in Fig 6 indicate that Case 2 exhibits the most favorable performance overall. The minimum temperature recorded is 24.9 degrees Celsius, the average temperature is 25.3 degrees Celsius, and the maximum temperature is 26.7 degrees Celsius. It provides the most effective air circulation among the configurations and optimally cools the room. The air conditioning units were positioned at the front of the classroom and on each of the side walls. There is a cross-flow of cooled air and effective mixing. This facilitates enhanced convective cooling. Consequently, the room experiences enhanced cooling.

In Case 3, the three air conditioning units were positioned equidistantly beneath the roof. The minimum temperature observed in the contour is degrees Celsius, the average temperature is 25.86 degrees Celsius, and the maximum temperature is 33.57 degrees Celsius. The air conditioning units were positioned equidistantly from one another to ensure uniform cooling across the space. However, there is a reduced circulation of cooled air throughout the system. There is an absence of disturbance in the room. Turbulence would enhance the cooling efficiency of an air conditioning system. The system operates less effectively than the other configurations.

Finally, in Case 4, a cold zone is situated at the center of the room. This is due to the presence of an air conditioning unit in the center of the left wall, along with two more units at the front and rear sides. The cooled air converges at the center of the room. The temperature is reduced in the center. It was lower than in case 3.

3.2.Velocity distribution

The velocity of the cooled air emanating from the air conditioning unit is pivotal in system cooling. Ensuring turbulent air mixing throughout the system will reduce the total room temperature. In Case 1, there is reduced air mixing. This is due to the increased distance between the air conditioning units. However, in scenario two, the mixing is superior owing to the cross-flow configuration among the three air conditioners. In scenario three, there is an absence of crossflow activity. The turbulence results in an elevated temperature. However, in Case 4, evidence of vortex formation is observable due to the airflow from three air conditioners converging in the center. The vortices facilitate effective mixing of cooled air in the system's central region. Consequently, this configuration provides superior cooling.

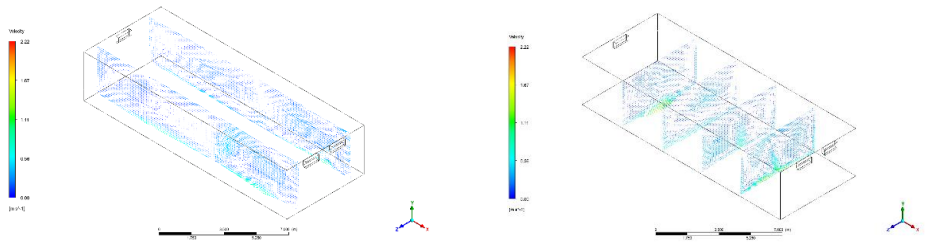


Fig. 7. Case 1 Air velocity distribution in horizontal plane(left) and vertical plane(right)

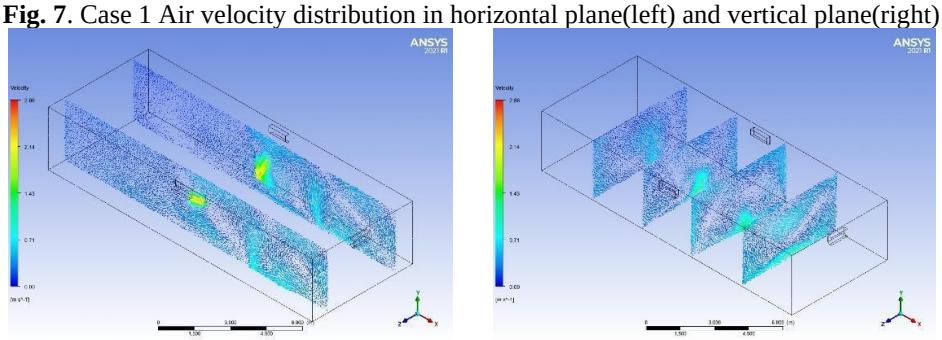


Fig. 8. Case 2 Air velocity distribution in horizontal plane(left) and vertical plane(right)

Table 3. Improvements in COP

Comparison	% increase COP
Case 1 to Case 2	17.92
Case 1 to Case 3	23.01
Case 1 to Case 4	7.36

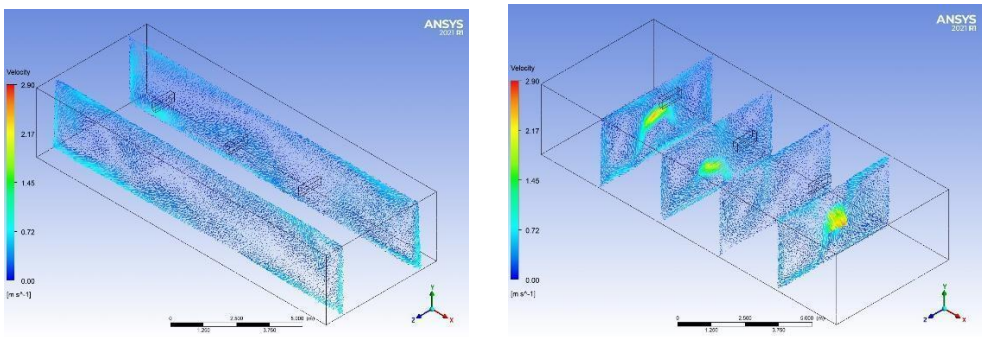


Fig. 9. Case 3 air velocity distribution in horizontal plane(left) and vertical plane(right)

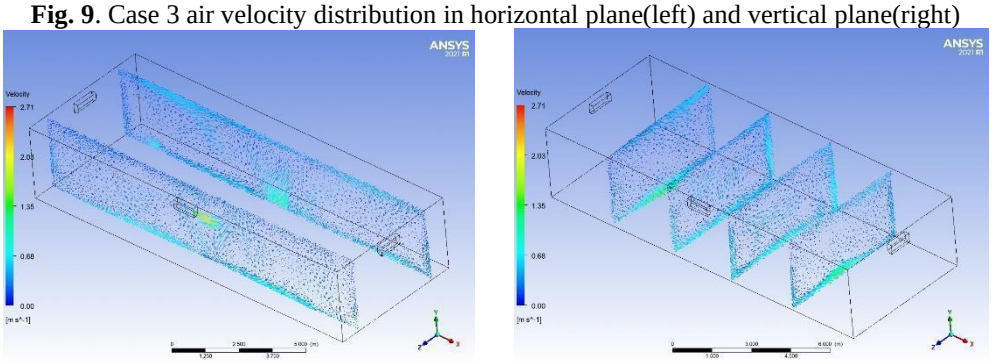


Fig. 10. Case 4 air velocity distribution in horizontal plane(left) and vertical plane(right)

4. Conclusion

This study has conducted a thorough analysis of the effects of various air conditioning setups on room cooling efficiency. The results underscore the essential need of air mixing and turbulence in attaining maximum cooling efficiency. In Case 1, the large distance between air conditioning units led to diminished air mixing and decreased cooling efficiency. Case 2 exhibited greater performance with a cross-flow design among three AC units, resulting in improved air mixing and good cooling efficiency. Case 3 demonstrated that the lack of crossflow activity and turbulence resulted in increased temperatures and diminished cooling efficiency. Ultimately, Case 4 demonstrated superior cooling efficiency attributable to the generation of vortices resulting from the convergence of airflow from three air conditioning units, hence promoting efficient mixing of cooled air in the middle area. The results indicate that air circulation and temperature distribution are significantly influenced by the positioning of air conditioning units. Among the configurations evaluated, Case 2, including AC units installed on the front and side walls, yielded the most efficient cooling performance, sustaining an average temperature of 25.3°C and facilitating effective air mixing through cross-flow. Case 4 demonstrated effective cooling, with prominent vortex production that improved air mixing at the center of the room. Nonetheless, Case 1 and Case 3 exhibited suboptimal outcomes owing to limited air mixing and insufficient turbulence, resulting in uneven cooling.

5. References

[1] X. Zhong, M. Cai, Z. Wang, Z. Zhang, and R. Zhang, “Influences of Heat Rejection from Split A/C Conditioners on Mixed-Mode Buildings: Energy Use and Indoor Air Pollution Exposure Analysis,” Buildings, vol. 14, no. 2, p. 318, Jan. 2024, doi: 10.3390/buildings14020318.

[2] Rampravesh Yadav1, Shivendra Singh2, Dr. B. Suresh3 “Investigation the Advisable Position of Split Air Conditioning Unit on Classroom Using Computational Fluid Dynamics (CFD)” International Journal of Innovations In Science Engineering And Management, vol. 3, no. 3, 2024, pp. 32–43.

[3] A. Patel, P. S. Dhakar, and JETIR, “CFD analysis of air conditioning in room using ANSYS Fluent,” journal-article, Nov. 2018. doi: 10.13140/RG.2.2.13462.50249.

[4] Y. H. Irawan, M. Abdulkadir, D. Sugati, Y. A. Jayatun, Harianto, and D. R. Hartana, “Investigating the Temperature and Air Velocity Distribution of Split-Type Air Conditioners Using Computational Fluid Dynamics,” conference-proceeding, 2018.

[5] T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu, “A NEW kt EDDY VISCOSITY MODEL FOR HIGH :REYNOLDS NUMBER TURBULENT FLOWS,” 1995.

[6] W. RODI, “Experience with two-layer models combining the k-epsilon model with a one-equation model near the wall,” American Institute of Aeronautics and Astronautics (AIAA), Jan. 1991. doi: 10.2514/6.1991-216.

[7] J. Axerio-Cilies, E. Issakhanian, J. Jimenez, and G. Iaccarino, “An aerodynamic investigation of an isolated stationary formula 1 wheel assembly,” Journal of Fluids Engineering, Transactions of the ASME, vol. 134, no. 2, 2012, doi: 10.1115/1.4005768.