FLUID DYNAMICS ANALYSIS OF A COMPUTER FAN FOR ENHANCED COOLING EFFICIENCY

Damora Rhakasywi
Mechanical Engineering
Universitas Pembangunan Nasional ”Veteran” Jakarta
Jakarta Selatan, Indonesia
rhakasywi@upnvj.ac.id

Rayhan Suryo Kusumo
Mechanical Engineering
Universitas Pembangunan Nasional ”Veteran” Jakarta
Jakarta Selatan, Indonesia

Angga Wijaya Taufik
Mechanical Engineering
Universitas Pembangunan Nasional ”Veteran” Jakarta
Jakarta Selatan, Indonesia

***Abstract*—Modern computers generate significant heat during operation, necessitating effective cooling methods to prevent damage and ensure longevity. Among these methods, the computer fan is widely used, circulating air around heat-producing components. This study delves into the factors influencing the cooling efficiency of computer fans. By harnessing CFD analysis, this study identifies key factors affecting cooling performance, offering valuable knowledge in computer cooling technology. Results reveal an iterative process yielding a progressively uniform fluid velocity. The fan blade center exhibits the highest velocity (1089.32 cm/s), forming a distinct spiral airflow pattern. Higher fan density (1.15 g/cm3) results in increased airflow due to a strengthened thrust force. Air velocity peaks approximately 6 cm from the fan center, diminishing with distance. Insights from this study contribute to designing more efficient computer fans, crucial for optimizing cooling efficiency while considering factors like density, velocity distribution, and noise levels.**

***Keywords*—Computational Fluid Dynamics, Computer Fan, Cooling Efficiency.**

I. Introduction

Modern computers generate significant heat, which can cause computer components to overheat and become damaged. Computer cooling is a critical factor in ensuring the performance and longevity of computers. The computer fan is one of the most commonly used cooling methods. The fan works by circulating air around the computer components that produce heat. The air moved by the fan helps transfer heat from computer components to the environment. The cooling performance of a computer fan is influenced by various factors, including the fan's shape, speed, and distance from computer components. To enhance the cooling performance of a computer fan, an analysis is needed to identify the factors affecting its cooling efficiency.

Computational Fluid Dynamics (CFD) analysis is one method that can be used to analyze airflow around a computer fan. CFD is a numerical method used to solve equations that describe fluid dynamics. This research utilizes CFD to analyze airflow around a computer fan. The aim of this study is to identify the factors influencing the cooling performance of the computer fan. The results of this research are expected to be used in designing a more efficient computer fan to improve cooling efficiency.

II. Methods

A. Computational Fluid Dynamic

The analysis of a computer fan using fluid dynamics methods to enhance cooling efficiency involves utilizing Computational Fluid Dynamics (CFD) techniques. This approach allows for a comprehensive examination of the airflow around the computer fan, considering factors such as fan design, rotational speed, and distance from heat-generating components. The goal is to identify the key elements influencing the cooling performance of the computer fan. This study employs the software Autodesk Inventor CFD 2021 for simulations and analyses, aiming to contribute insights for designing a more efficient computer fan and ultimately improving overall cooling efficiency.

B. Research Stages

In the research of computer fan analysis using fluid dynamics methods to improve cooling efficiency, there are three main stages that must be passed, namely: preprocessing, solving or processing, and postprocessing. The numerical method approach, with the help of the software Autodesk Inventor CFD 2021. This research also includes a three-dimensional (3D) representation of the computer fan, adding a visual dimension for a deeper understanding of the dynamics of airflow and cooling efficiency.

1) Preprocessing: Preprocessing serves as the initial step in constructing and analyzing a computational fluid dynamics (CFD) model. This phase encompasses several sub-steps, including geometry creation, domain determination, meshing creation, and specification of the parameters to be utilized.

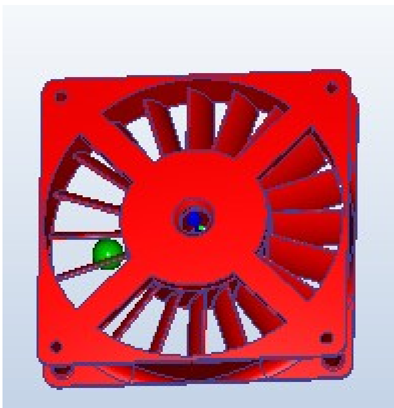

Fig. 1. Computer Fan 3D Design.

2) Processing or Solving: Utilizing Autodesk Inventor CFD 2021 software, the conditions established during the preprocessing stage within the software will be computed (iterated). If convergence criteria are met, the process proceeds to postprocessing; if not, the stage regresses to the meshing creation phase or by adjusting the convergence criteria.

3) Postprocessing: Postprocessing in Autodesk Inventor CFD 2021 software involves presenting and analyzing the obtained results, including both qualitative and quantitative data. These data encompass iterations, velocity magnitude, density, and velocity components (Vx, Vy, Vz).

III. Results

A. Iteration

Figure 2 illustrates the results of fluid flow simulation. The graph reflects fluid velocity at a specific point in space at a given time, generated through an iterative method. In the first iteration, the initial fluid velocity is calculated based on the initial conditions and is then used to compute the fluid's pressure and temperature. This process continues until the fluid's velocity in the next iteration shows no significant change from the preceding iteration. In the provided figure, the first iteration is represented by the blue line, indicating fluid velocity at 0.1 seconds. Subsequent iterations are depicted by the red line, reflecting fluid velocity at 0.2 seconds. Initially, the fluid velocity is non-uniform, but it becomes more uniform in the following iteration as it takes into account the velocity from the previous iteration.

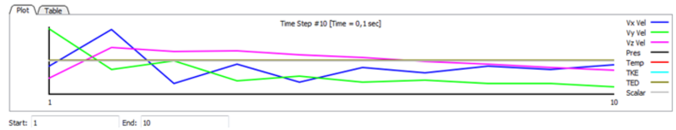

Fig. 2. Iteration of Computer Fan Simulation.

B. Velocity Magnitude

The simulation results in Figure 3 indicate the speed of air movement around the fan. The highest value, 1089.32 cm/s, signifies the fastest air movement at the center of the fan blades. Lower values indicate slower air movement outside the blades. These values also reveal the airflow pattern around the fan, where the air moves from the center of the blades outward, forming a spiral pattern.

C. Density

The density, as seen in Figure 4 at 1.15 g/cm3, tends to result in a larger airflow. This is attributed to the denser computer fan having a greater mass, generating a stronger thrust force and consequently a more robust airflow. The fan size, also evident in the figure with a diameter of 6.06714 inches, is capable of producing a larger airflow as larger blades can push more air.

D. Velocity (Vx, Vy, Vz)

1) Vx-Velocity: The graph in Figure 5 illustrates that the maximum air flow velocity is attained at a distance of approximately 6 cm from the center of the fan. Subsequently, the air flow velocity decreases as the distance from the center of the fan increases.

2) Vy-Velocity: The analysis results in Figure 6 indicate a significant increase in air velocity around the computer fan blades as the distance from the center of the blade decreases. The highest velocity, approximately 15 cm/s, is observed at the blade's edge at a distance of 2 cm from the center, then gradually decreases to 5 cm/s at a distance of 10 cm.

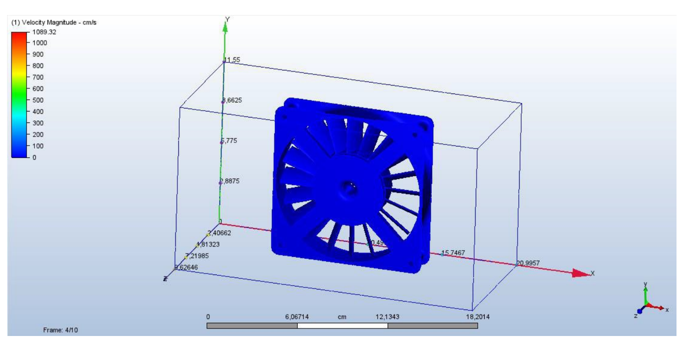

Fig. 3. Computer Fan Simulation Velocity Magnitude.

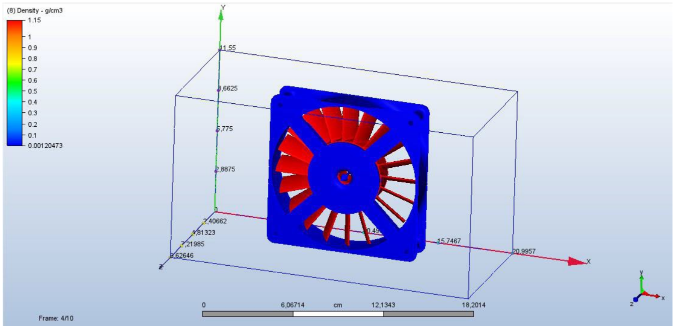

Fig. 4. Simulation Density of Computer Fan.

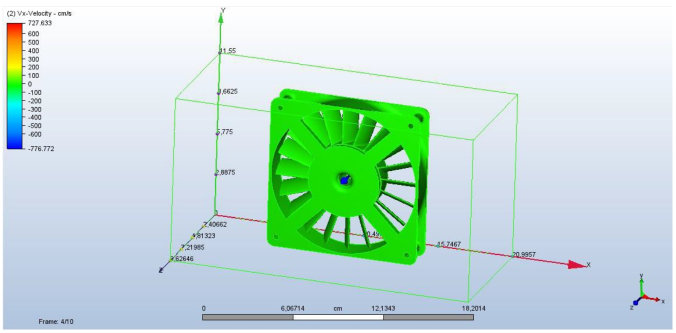

Fig. 5. Represents the Vx-Velocity Simulation of a Computer Fan.

3) Vz-Velocity: The analysis results in Figure 7 illustrate the air velocities at various points within the fan system. The recorded air inlet speed is 5 m/s, depicting the velocity as air enters the fan. As the air traverses the fan blades, its speed increases, reaching a maximum of approximately 10 m/s around the blades. Subsequently, the air speed decreases as it moves through the back of the fan, ultimately exiting the fan at a speed of 15 m/s. High air velocities contribute to enhanced airflow and cooling efficiency; however, it is crucial to balance this with the potential noise issues associated with excessively high speeds.

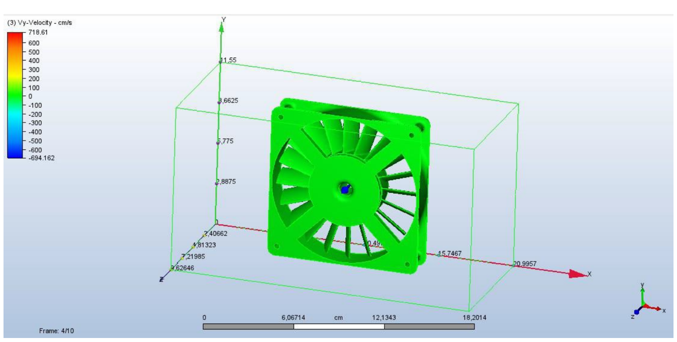

Fig. 6. Represents the Vy-Velocity Simulation of a Computer Fan.

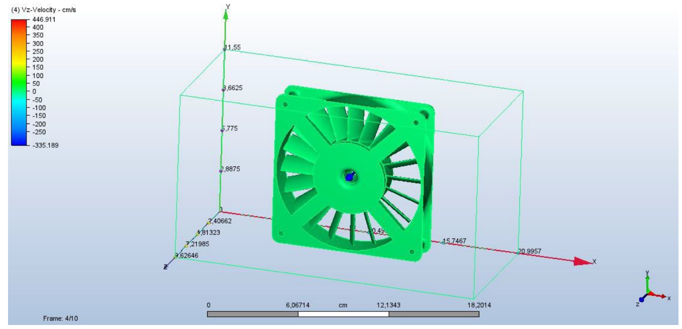

Fig. 7. Represents the Vz-Velocity Simulation of a Computer Fan.

IV. Conclusions

In the simulation of computer fan fluid flow, an iterative method is employed to achieve uniform fluid velocity. The results indicate the highest velocity at the center of the fan blades, reaching a peak value of 1089.32 cm/s, forming a spiral pattern of airflow. The higher fan density (1.15 g/cm3) results in a larger airflow due to a stronger thrust force. The distribution of air velocity shows a peak velocity approximately 6 cm from the center of the fan, decreasing with increasing distance. Overall, the simulation results provide insights into the fluid dynamics and characteristics of the computer fan, crucial for enhancing cooling efficiency while considering factors such as density, velocity distribution, and noise levels.

References

[1] Anggrana, B. G. D., Karohika, I. M. G., S.T., M.T. Analisis Aerodinamika Bodi Mobil dengan Variasi Kecepatan Menggunakan Perangkat Lunak CFD. Sibatik Journal, 2022; 1: 1456- 1460.

[2] Kapilan, N., Manjunath Gowda, M., Manjunath, H. N. Computational Fluid Dynamics Analysis of an Evaporative Cooling System. Journal of Mechanical Engineering. 2016; 66: 119-123.

[3] Syafiul, A., Utina, M. R. Analisa Pengaruh Variasi Kecepatan Terhadap Tekanan pada Model Kapal Selam dengan Menggunakan Simulasi Numerik. Jurnal Wave. 2017; 11: 64- 66.

[4] Al-Kindi, H., Purwanto, Y. A., Wulandani, D. Distribution Analysis Hot Air Flow of Rack Type Dryer With Energy Source From Exhaust Gas Using Computational Fluid Dynamics (CFD). Jurnal Keteknikan Pertanian. 2015; 3: 11-12.

[5] Yang, S., Ai, Z., Zhang, C., Dong, S., Ouyang, X., Liu, R., Zhang, P. Study on Optimization of Tunnel Ventilation Flow Field in Long Tunnel Based on CFD Computer Simulation Technology. Journal of Sustainability Science and Engineering. 2022; 14(18): 4-14.