[1,*]Wenpan Dai
[2]Wensheng Yu
[3]Jiawei Wang
[4]Jinhui Tang

Optimization Based on Computer Fluid Dynamics Finite Element Method Automotive Cooling Fan Structure

Journal of Electrical Systems

Abstract: - With the continuous improvement of vehicle comprehensive technology, the heat load generated by automotive engine systems, hydraulic systems, and other devices also increases. Therefore, improving the heat dissipation efficiency of the cooling system is a key technology that cannot be ignored in the current vehicle development process. The car cooling fan is an important component of the cooling system, and its working characteristics will directly affect the cooling effect of the engine. This article uses the finite volume method of computer fluid dynamics to optimize the structure of a heat dissipation fan, analyze and define the flow field control volume required for its numerical calculation, use structured and unstructured grids to discretize the internal flow field region, select pressure inlet and pressure outlet boundary conditions, and simulate and analyze the fluid flow in the radiator based on fluid dynamics theory calculations. The orthogonal experimental method was used to analyze various performance indicators during the experimental process. The results showed that the comprehensive performance of the No. 3 optimized cooling fan was the best, with a 10.45% increase in static pressure efficiency, a 14.03% increase in dynamic pressure efficiency, and an 11.26% increase in maximum flow velocity. The optimized fan flow field streamline became more full and uniform, significantly improving the fluid dynamics performance of the air in the cooling system.

Keywords: Computer Technology, Finite Volume Method, Fluid Mechanics, Fluent Analysis, Orthogonal Test Method, Static Pressure Efficiency, Dynamic Pressure Efficiency.

I. INTRODUCTION

At present, the design and performance evaluation of cooling fans are mostly based on air duct tests. But in recent years, with the rapid development of computer technology and computational fluid dynamics technology, using commercial CFD software for simulation performance testing of cooling fans will be a more convenient and economical testing method. Many related studies have also shown that its calculation results have a certain degree of reliability [1,2]. Therefore, the commercial CFD software Fluent was used to simulate the overall flow field of the preliminarily designed cooling fan, obtaining the working characteristics of the cooling fan, providing a basis for corresponding experimental testing, and laying a foundation for structural optimization. Computer simulation has the characteristics of high efficiency, low cost, and repeatability. Computer simulation can avoid a large amount of experimental costs and limitations, and improve the efficiency and accuracy of research. In addition, computer simulation can provide more detailed data and information, helping researchers better understand fluid dynamics problems [3].

The structural design of the automobile cooling system is generally based on the need to meet the best heat dissipation efficiency, but the working environment of the heat dissipation device is often in an overheated state, which is inconducive to the normal and efficient automobile operation [4-6]. In order to gain good dynamic property, economy, emission and safety, modern automobiles have implemented a large number of new technologies, such as exhaust gas turbocharging systems, high-pressure cylinder direct injection technology, twin-vortex turbo technology, fast engine start-stop technology, as well as emission system purification technology, etc. These systems increase the automobile thermal load to a certain extent and reduce the automobile heat dissipation efficiency which creates many problems in automobile heat dissipation and also becomes one focus of domestic and foreign researchers [7-9]. According to relevant literature, since 1993, International Conference of the American Society of Automotive Engineers on Vehicle Thermal Management Systems is held once every two years. John discussed some issues worthy of attention in the design of folding bikes and Safi cooling systems [10]. Ohshima et al. tested the effect of cooling air outlet on automobile aerodynamic performance and cooling performance through experiments on model wind tunnel, automobile wind tunnel and climate wind tunnel [11]. Some European and American scientific research institutions have also

[1] College of Transportation and Navigation, Quanzhou Normal University, Quanzhou, China
[2] College of Transportation and Navigation, Quanzhou Normal University, Quanzhou, China
[3] College of Transportation and Navigation, Quanzhou Normal University, Quanzhou, China
[4] College of Transportation and Navigation, Quanzhou Normal University, Quanzhou, China
*Corresponding author: Wenpan Dai

developed a variety of new cooling system technologies, such as the advanced engine cooling system proposed by Valeo, the Intelligent Cooling System, Adaptive Cooling system, and Active cooling system, etc. from THEMISTM and DANA in the United States [12-17]. Seen from the above literature, there is need to enable good automobile heat dissipation effect without affecting the automobile performance at the same time. As far as the current application research is concerned, cooling fan is an important part of the automobile cooling system, and the fan working characteristics directly affect the heat dissipation effect of the engine. Therefore, good structure design of cooling fan in the automobile cooling system helps improve the experimental cooling effect. Improperly designed fan structure parameters will lead to poor engine cooling, resulting in deteriorated working environment of the engine, which in turn affects the engine performance and service life. At present, the industry mainly reduces the energy consumption and improves the performance of cooling fans through structural optimization [18].

With the existing automobile cooling fan structure in the laboratory as the prototype, the paper uses the "Pro-E" software to model its structure, refers to the data design requirements such as static pressure level and flow rate level, and combines modern computer technology to optimize structural parameters of the cooling fan blades based on the finite volume method. "Fluent" software is used to simulate and analyze various hydrodynamics parameters of the optimized structure, so as to find the optimal solution to meet the heat dissipation performance requirements of the cooling fan while considering the balanced matching relationship between heat dissipation and power.

II. RESEARCH METHOD AND DESIGN YHEORY OF COOLING FAN

A. Cooling Fan Research Method

However, technology has entered a stage of rapid development, with computer analysis capability rapidly improved. The creation, modification and analysis through computer modeling have replaced the traditional sample mold testing process. Although these fluid mechanics calculation software can lead to fan performance data, air duct test is still required to compare the air duct experimental data with the simulation data, so that it is possible to judge whether the simulation results are accurate [19]. The process is shown in Figure 1.

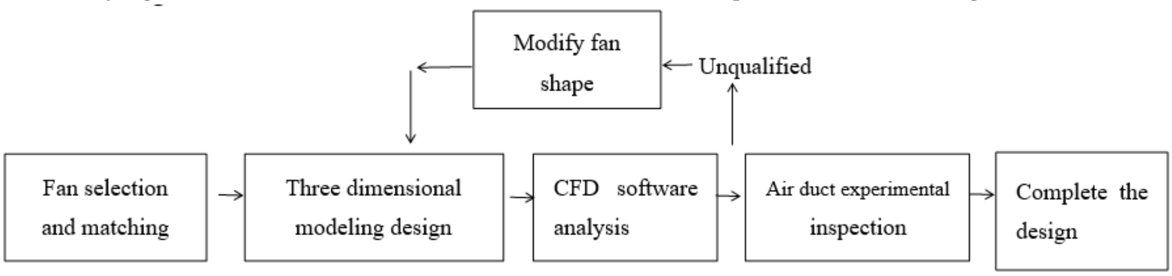


Figure 1: Computer-aided Design Process of Engine Cooling Fan

B. Structural Parameters of Cooling Fan

The most common structural parameters of fans include blade number, the maximum blade thickness, blade length, blade width, the maximum blade diameter, blade installation angle, fan outer diameter, hub diameter, and hub thickness. As a connecting part of the fan blades, the hub is in the shape of a thin-walled cylinder. The circle area in the front view is the hub area. DL is the hub diameter; Lb is the blade length. Dw is the fan outer diameter. The ratio of hub diameter to the fan outer diameter is called the hub ratio [20]. The top view of the fan blade structure is shown in Figure 2, and the front view of the fan structure is shown in Figure 3.

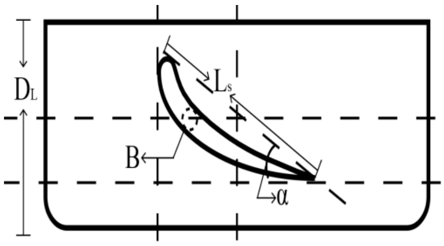


Figure 2: Top View of Blade Structure

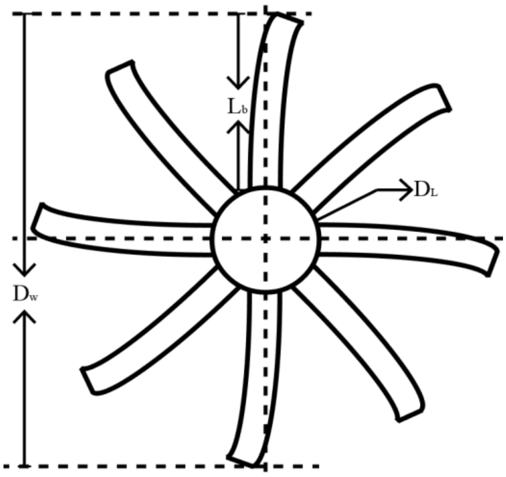


Figure 3: Front View of Fan Structure

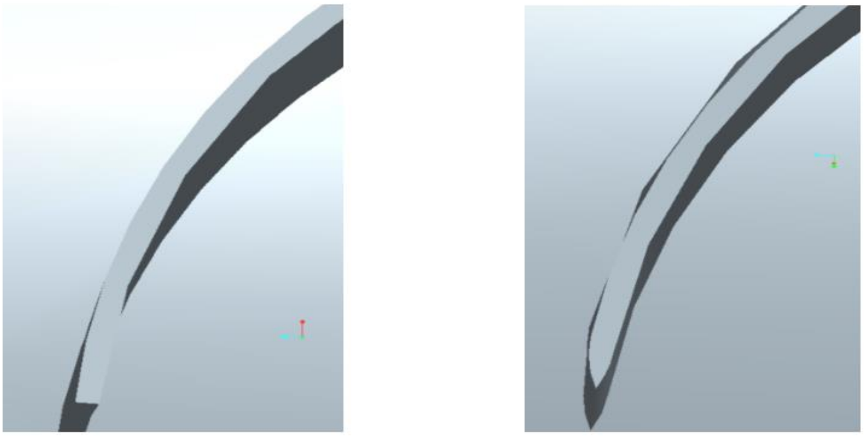
C. Variable Parameter Design of Cooling Fan

The dimensions of the original graphics demanding analysis and optimization are as follows, rated electric power 120W, fan outer diameter 308mm, hub diameter 106mm, blade length 101mm, blade tip width 29mm, blade root width projection 21mm, blade safety angle 28.3°, blade number 10, hub thickness 43mm. The parameters are shown in Table 1.

Table 1: Structural Parameters of the Original Fan

Parameter name	Hub ratio	Outer diameter	Hub diameter	Hub thickness	Blade length	Blade root X-axis projection	Blade root Y-axis projection	Blade installation angle
Value	0.35	308mm	106mm	43mm	101mm	39mm	21mm	28.3°

The original fan is optimized with reference to the design standard, and the 3D modeling graphics are subjected to parametric modeling via Pro-E. When the blade number is odd, the fan is more stable in rotation, so the optimized fans are all 9-blade fans [21]. Moreover, the blade side structure of the original fan is flat, which increases the resistance when the blade cuts the airflow, and also creates more turbulent airflow swirls, which may produce noise. Therefore, all optimized graphs are designed with blade structure, as shown in Figure 4(a), (b).



a) Blade structure with flat sides b) Blade structure with blades on sides

Figure 4: Side Structure of the Fan Blade before and after Optimization

Since the two parameters of the hub ratio i and the blade installation angle α greatly affect the cooling performance, it is necessary to change these two parameters to optimize the fan structure, reduce energy consumption, and increase heat dissipation efficiency. First, the structural design takes the original hub ratio i and blade installation angle α as the basic parameters to change the basic parameters. Then, the orthogonal test method is used to analyze and compare the performance under different parameter combinations, and the optimal solution in the experiment is obtained, so that a new fan structure model can be designed according to the optimal solution parameters. Compared with the original fan's hub ratio of 0.35 and blade installation angle of 28.3°, the hub ratio is set in units of 0.05 to set three parameters as 0.30, 0.35 and 0.4, and the blade installation angle of

28.3° is set in units of 2°to set parameters as 26.3°, 28.3° and 30.3°. The orthogonal test table is designed as shown in Table 2.

Table 2: Orthogonal Test Method

Fan Group	Hub Ratio i	Blade Installation Angle α
No. 1 Fan	0.30	26.3°
No. 2 Fan	0.30	28.3°
No. 3 Fan	0.30	30.3°
No. 4 Fan	0.35	26.3°
No. 5 original fan	0.35	28.3°
No. 6 Fan	0.35	30.3°
No. 7 Fan	0.40	26.3°
No. 8 Fan	0.40	28.3°
No. 9 Fan	0.40	30.3°

The radiator fan is modeled according to the nine sets of data in the orthogonal test table, and the nine sets of parameter modeling diagrams are shown in Figure 5.

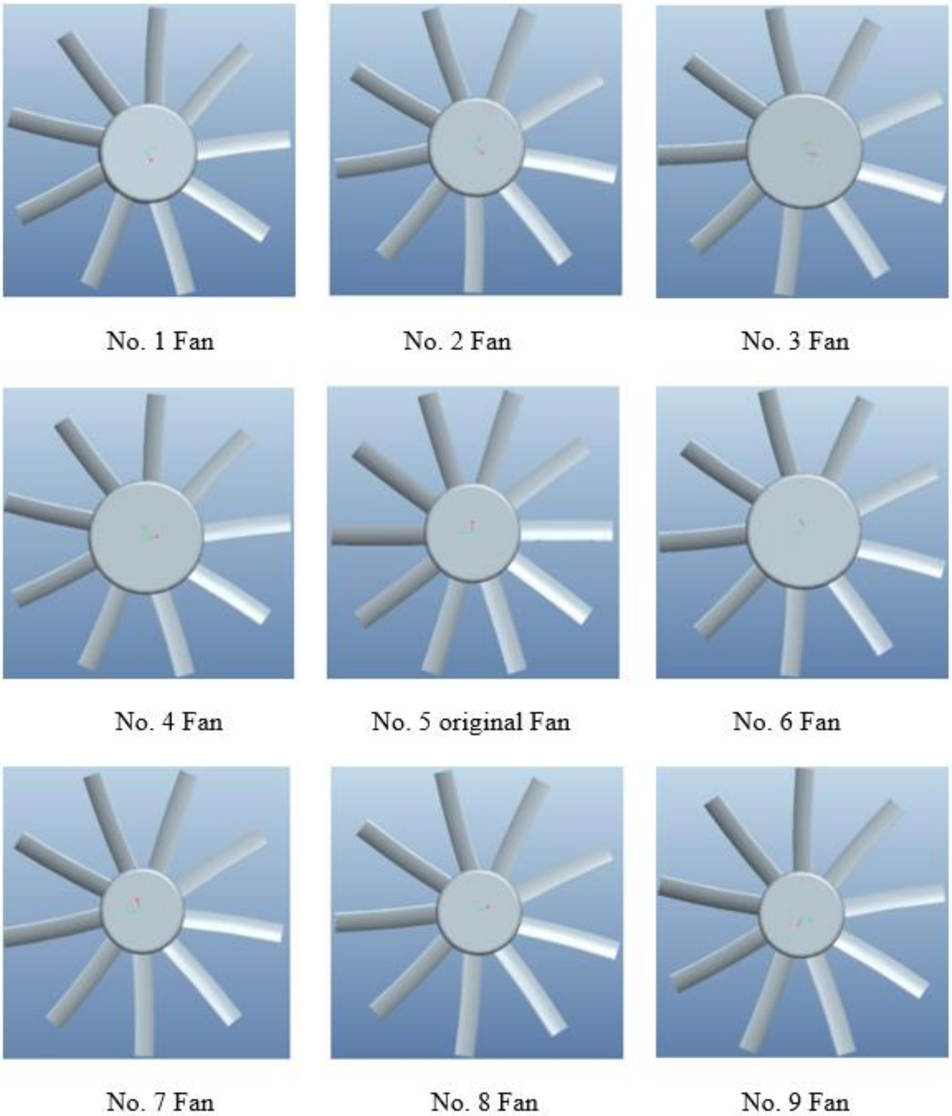


Figure 5: Nine sets of Modeling Diagrams According to Orthogonal Test Method

III. Flow Field Simulation Analysis

A. Flow Area Establishment

First, in establishment of the rotating flow area, the fan main body should be wrapped, and the rotating flow area should not be smaller than the fan main body. A cylindrical rotating flow area with a base area of about 79382.34mm2 and a height of 49mm was established. An outer cylinder flow area is set with a bottom area of about 297872.96mm2, a length of the air inlet area of 1000mm, a length of the air outlet area of 1000mm, and a total length of 2000mm, as shown in Figure 6.

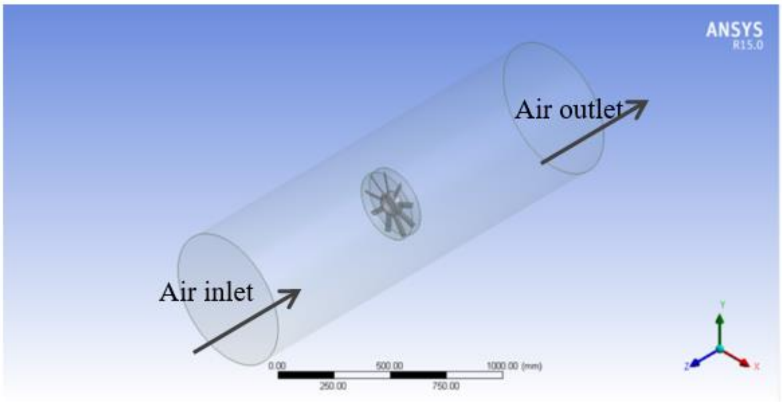


Figure 6: Establishment of Internal and External Flow Area

Finally, we need to set the air inlet, air outlet, and flow area boundary. The inlet is named "inlet", the outlet is named "outlet", and the flow area surface is named "wall".

B. Grid Division

Due to the bending of the fan blade, there is complex interface between the blade and the hub. Hence, a tetrahedral grid is used in division. The outer flow area and the rotating flow area need to be divided separately because of their different degrees of accuracy. First, the outer flow area is divided. The grid of the outer flow area is divided in unit of 15mm, and the grid of the rotating flow area is divided in unit of 5mm. A total of 2,499,627 grids are divided. Figure 7 shows the grid division results of the outer flow area, and Figure 8 shows the analysis results of the rotating flow area.

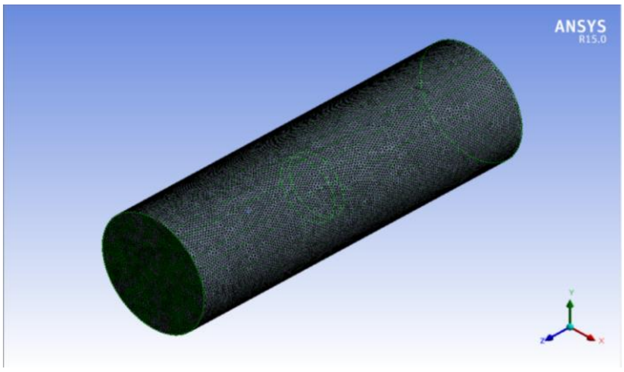


Figure 7: Grid Division Diagram of Outer Flow Area

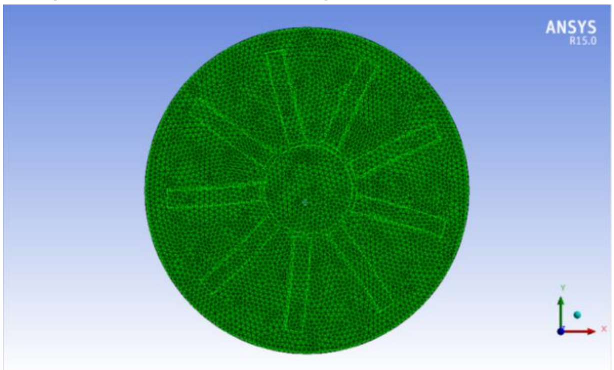


Figure 8: Grid Division Diagram of Rotating Flow Area

C. Setting of the Turbulence Model

When the simulation begins, the calculation process function needs to converge or show "solution is converged", indicating that the analysis is over. As shown in Figure 9, when the function is iteratively calculated for 876 times, the function image converges. Convergence has been achieved before the original setting of 1000 iterative computations is completed, indicating that the simulation is successful, and convergence also occurs in the second cycle iteration, indicating that the simulation results are credible.

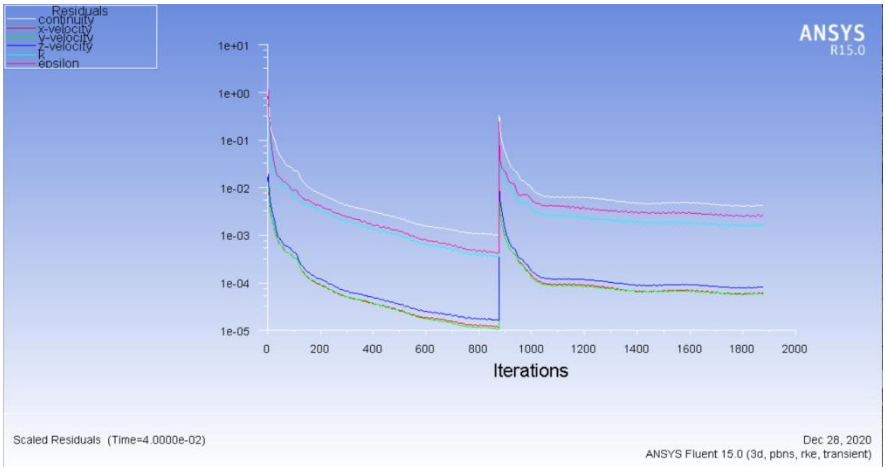


Figure 9: Simulation Calculation Convergence

The average static pressure, average dynamic pressure, and average flow velocity at the inlet and outlet are selected from the export data in "Report". The results are checked to plot the wind speed streamline diagram, as shown in Figure 10.

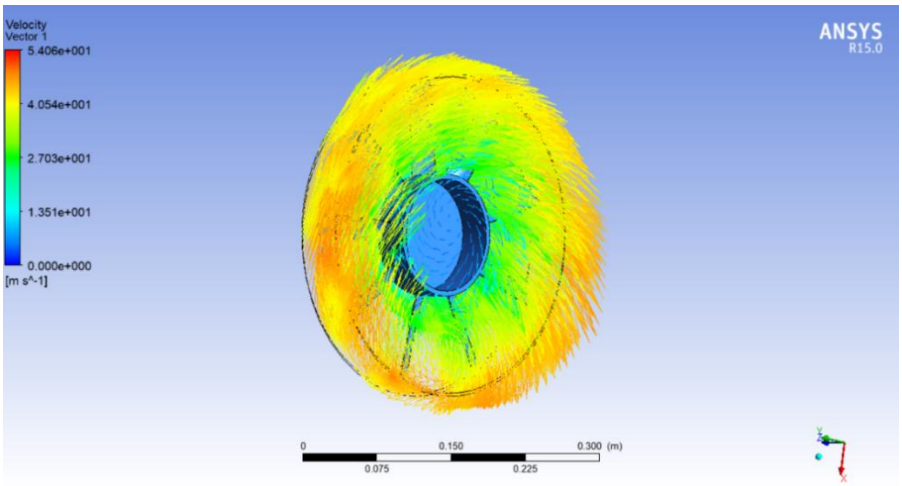


Figure 10: Wind Speed Streamline Diagram of the Rotating Area

D. Simulation Results

After the analysis setting according to the same simulation parameters, the data was checked in the above-mentioned way, and comparison was made for the static pressure of the air inlet, the static pressure of the air outlet, the dynamic pressure of the air inlet, the dynamic pressure of the air outlet, the average flow velocity of the air inlet, the torque, and the maximum sound power level. The data are summarized as shown in Table 3.

Table 3: Export Data of Each Fan

Cooling fan model	static pressure of the air inlet /Pa	static pressure of the air outlet /Pa	dynamic pressure of the air inlet /Pa	dynamic pressure of the air outlet /Pa	average flow velocity of the air inlet /m/s	maximum sound power /dB	torque /N·m
No. 1 Fan	0.04	30.40	30.41	50.98	7.32	110.37	0.67
No. 2 Fan	0.05	32.46	32.46	56.56	7.87	112.99	
No. 3 Fan	0.48	34.98	34.98	61.37	8.15	106.82	
No. 4 Fan	0.05	32.46	32.46	56.56	7.87	112.99	
No. 5 original fan	0.03	34.01	34.01	58.47	7.45	114.29	
No. 6 Fan	0.04	34.04	34.04	60.54	7.46	108.21	
No. 7 Fan	0.05	30.01	30.52	51.25	7.58	107.92	
No. 8 Fan	0.03	31.95	31.95	54.23	7.72	113.33	
No. 9 Fan	0.02	33.89	33.89	59.79	8.04	100.86	

E. Comparison between Calculation Results of Cooling Performance Parameters

The data in Table 3 were substituted into the formula, and the calculation process takes the original fan calculation as an example [22].

Static pressure: $p_{sf} = p_{sf2}\text{-}p_{sf1}$ (1)

Dynamic pressure: $p_{df} = p_{df2}\text{-}p_{df1}$ (2)

Full pressure: $p_{tf} = p_{sf}\text{-}p_{df}$ (3)

Volume flow rate: $q_{Vx} = v \cdot A_2$ (4)

Mechanical input power of the motor: $P_0 = \frac{N \cdot T_f}{9550}$ (5)

Static pressure power: $P_{es} = q_{Vx} \cdot p_{sf}$ (6)

Full voltage power: $P_{et} = q_{Vx} \cdot p_{tf}$ (7)

Static pressure efficiency: $\eta_{sf} = \frac{P_{es}}{P_0} \times 100\%$ (8)

Dynamic pressure efficiency: $\eta_{tf} = \frac{P_{et}}{P_0} \times 100\%$ (9)

The nine groups of data are calculated according to the above formula to derive the data as shown in Table 4.

Table 4: Calculation Results of the Nine Groups of Simulation Tests

Fan Model	Static Pressure/Pa	Full pressure/Pa	Static Pressure Efficiency	Dynamic Pressure Efficiency
No. 1 Fan	30.41	51.55	24.37%	41.30%
No. 2 Fan	32.41	56.51	26.97%	47.03%
No. 3 Fan	34.50	60.89	29.79%	52.57%
No. 4 Fan	30.36	50.93	23.38%	39.23%
No. 5 original fan	33.98	58.44	26.80%	46.10%
No. 6 Fan	34.00	64.50	26.82%	50.88%
No. 7 Fan	29.96	50.69	24.01%	40.61%
No. 8 Fan	31.92	54.20	25.97%	44.10%
No. 9 Fan	33.87	59.77	28.82%	50.86%

By analyzing the calculated parameters and comparing the nine sets of data, it is found that No. 3 fan has a static pressure efficiency of 29.79%, a full pressure efficiency of 52.57%, both of which are the highest, while No. 4 fan has a static pressure efficiency of 23.38%, a full pressure efficiency of 39.23%, both of which are the lowest. Therefore, among the nine groups of fans, No. 3 fan has the best aerodynamic performance.

F. Comparative Analysis of Flow Field

According to the analysis data, No. 3 fan has the best cooling performance optimization effect among each simulated radiator cooling fan. The flow field diagram of No. 3 fan and No. 5 original fan is compared as shown in Figure 11 and Figure 12.

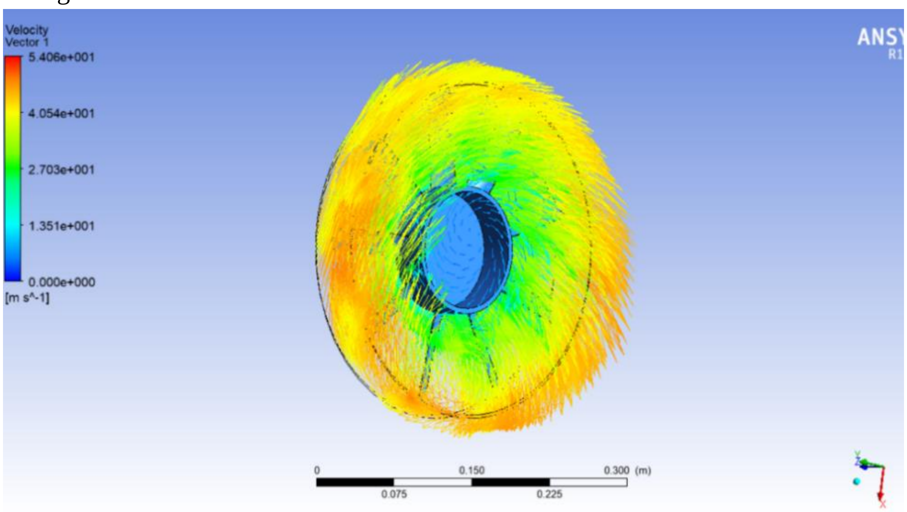


Figure 11: Flow Field Details of No. 5 Fan

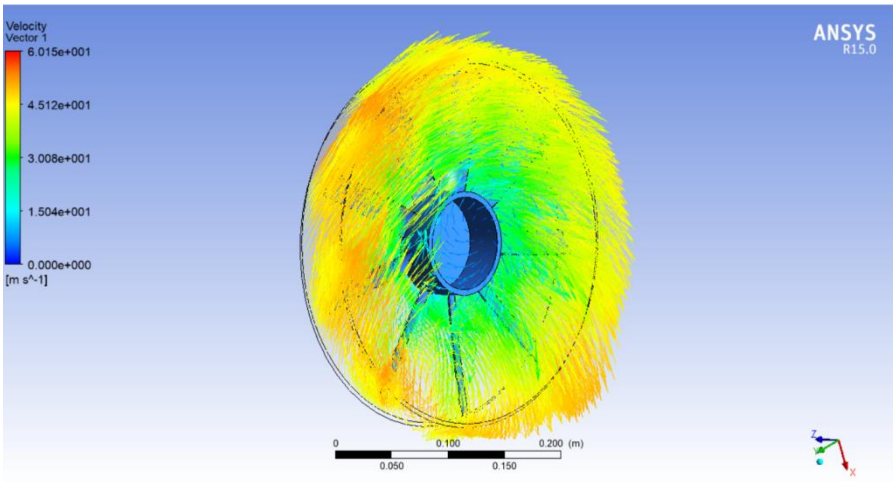


Figure 12: Flow Field Details of the No. 3 Original Fan

In the figure, bluer tip color indicates lower flow rate, and redder color indicates higher flow rate. It can be seen from the figure that No. 5 original fan has a maximum flow velocity of 54.06m/s, while No. 3 fan has a maximum flow velocity of 60.15m/s. It can be seen that the maximum flow velocity of the flow field has increased by 11.26%. The flow line of No. 3 looks fuller and more uniform, thus greatly reducing the impact of noise. Since the average flow velocity determines the fan performance and efficiency to a certain extent, No. 3 fan has superior performance than the original fan.

IV. Conclusions

Using modern computer technology, this paper models and optimizes the original vehicle cooling fan, and hydrodynamic performance of the fluid medium in the cooling system was analyzed using the "Fluent" software. In the experimental process, the orthogonal test method was used to design and analyze various performance indexes. Among the nine groups of simulated fans, No.3 fan has a static pressure efficiency of 29.79%, a full pressure efficiency of 52.57%, both of which are the highest. Where, the static pressure efficiency is increased by 10.45%, and the dynamic pressure efficiency is increased by 14.03%. Moreover, the maximum flow velocity of No. 3 fan is 60.15m/s, which is 11.26% higher than that of the original fan, indicating better aerodynamic performance. Therefore, it is finally determined that the hub ratio and blade installation angle of the radiator fan are designed to be 0.35 and 30.3°. The optimized fan flow field streamline is more plump and uniform, which greatly improves the hydrodynamic performance of the air in the cooling system, and provides a new idea for the overall structural design of the automobile radiator fan. The research on the vehicle cooling system in this paper is currently limited to the cooling fan structure, and the number of orthogonal tests needs to be increased in order to design the optimal structural parameters more accurately.

Acknowledgment

Natural Science Foundation of Fujian Province: Study on the establishment and operation characteristics of three evaporator organic Rankine cycle system for recovering ship waste heat.2022J011110.

References

[1] Wang QK, Chen K. Characteristics of the influence of variable circulation index on the aerodynamic performance of twisted blades in micro axial fans. Journal of Mechanical Engineering, 2009, 45 (04): 76-82.

[2] He Q, Mao JG, He XM. CFD analysis of the performance of automotive engine cooling fans. Automotive Technology, 2009 (01): 46-48.

[3] Ma ZW, Wang ZY, Wei JW et al. Computer simulation research on fluid dynamics problems in university physics College Physics, 2016, 35 (10): 17-19+34.

[4] Zhang HJ, translated by Li Renye. Vehicle Cooling System Design Manual. Beijing: National Defense Industry Press, 1984.

[5] Yao ZP, Wang XG. Vehicle Cooling Heat Transfer. Beijing: Beijing Institute of Technology Press, 2001.

[6] Frank Melzer, Ullrich Hesse, Gerta Rocklage and Manfred Schmitt. Thermomanagement. SAE Technical Paperss. 1999, 1:0238.

[7] Valaszkai L, Jouannet B. Cooling System Optimization for Euro4-Epa/02 Heavy-Duty Trucks. SAE Technical Paperss. 2000, 1: 0964.

[8] Robert D, Chalgren. Thermal comfort and engine warm-up optimization of a low-flow advanced thermal management system. SAE Technical Paperss. 2004, 1: 0047.

[9] Lee HS, Cholewczynski LW. A study on convetion and boiling heat-transfer modes in a standard engine cooling system. VTMS6. 2003: 805-821.

[10] John RC. The design of automobile and racing car cooling systems. SAE Technical Paperss. 1997: 971835.

[11] Ohshima T, Hamatani K, Ninoyu M et al. Influence of the cooling air flow outlet ontheaerodynamic characteristics. JSAE Review. 1998, 19(2): 137-142.

[12] Matthieu C, Alex J, Darren B et al. Valeo Engine Cooling. Advanced Engine Cooling Thermal System On a Dual Voltage 42V/14V Minivan. SAE Technical Paperss. 2001, 1: 1742.

[13] Matthieu C, Bertrand G et al. The Need for An Electrical Water Valve in a (THEMISTM). SAE Technical Paperss. 2003, 1: 0274.

[14] Revereault P, Gessier B, Chanfreau M. Intelligent vehicle system thermal management in a mild hybrid - diesel vehicle. VTMS6, 2003: 545-560.

[15] Robert DC. Thermal comfort and engine warm-up optimization of a low-flow advanced thermal management system. SAE Technical Paperss. 2004, 1:0047.

[16] Brian L, Gordon GP, John HJ, Oner A. Development of the Enhanced Vehicle and Engine Cooling System Simulation and Application to Active Cooling Control. SAE Technical Papersss. 2005, 1: 0697.

[17] Ngy-Srun A, Jouanny P, Potier M, Genoist J. Ultimate Cooling™ System for New Generation of Vehicle. SAE Technical Papersss. 2005: 1.

[18] Zhou J. The Timetable for China's Ban on the Sale of Fuel Vehicles Has Been Released, and Fuel Vehicles Should Be Fully Withdrawn in 2050. Business Culture. 2019, 18: 79-81.

[19] Cui N. A Review of the Research on the Fan Drive Mode of the Construction Machinery Cooling System. Mechanical Engineering & Automation. 2017, 2: 211-213.

[20] Zheng ZF, Zhu MT. Parameterized Design and Optimization of Engine Cooling Fan Blades. Machinery Design & Manufacture. 2020, 12: 74-77+82.

[21] Sun XF. Aerodynamic Characteristics Analysis and Blade Optimization Design of Automobile Engine Cooling Fan. Nanjing University of Aeronautics and Astronautics, 2019.

[22] Wang FJ. Computational fluid Dynamics Analysis-Principles and Application of CFD Software. Beijing: Tsinghua University Press. 2004: 26-57, 74-81, 185-253.