Journal of Precision Instrument and Machinery (2025)
Clausius Scientific Press, Canada
DOI: 10.23977/jpim.2025.050103
ISSN 2616-1990 Vol. 5 Num. 1

Application of Computational Fluid Dynamics in Heat Transfer Optimization of New Energy Motor Cooling Structures

Wang Weisa

High Mark Group, Beijing, China

Keywords: Computational Fluid Dynamics; New Energy Sources; Motor Cooling Structures; Heat Transfer Optimizations

Abstract: In the field of new - energy motors, the improvement of their performance is significantly restricted by insufficient heat dissipation efficiency. Traditional design methods often struggle to meet the thermal management requirements under high power density. Computational Fluid Dynamics (CFD) technology has the outstanding ability to accurately simulate the flow and heat transfer processes. Relying on this ability, it has opened up a new path for optimizing the heat dissipation structure. This research focuses on the characteristics of the internal heat source distribution in the motor. By combining with multi - physical field coupling analysis, an efficient numerical model is established to explore the parametric design and optimization strategy of the heat dissipation structure. After integrating the response surface method and the genetic algorithm, a composite heat dissipation scheme is proposed, aiming to break through the limitations of the traditional heat dissipation mode. Experimental verification shows that this method is remarkably effective in improving the heat dissipation performance and energy utilization efficiency, providing strong theoretical support for the lightweight and reliability design of new - energy motors.

1. Introduction

With the rapid development of new energy technologies, the motor, as the core power unit, is showing a trend of high power density and miniaturization, which poses extremely severe challenges to the heat dissipation system. Traditional heat dissipation design mainly relies on empirical formulas and simplified models, and it has difficulty accurately capturing the interaction between complex flow fields and temperature fields, resulting in relatively low thermal management efficiency. Computational fluid dynamics technology can clearly reveal the fluid - solid coupling mechanism through numerical simulation, thus making it possible to conduct refined heat source analysis and optimize the heat dissipation structure. However, most existing studies are limited to single - physical - field analysis and lack a systematic and in - depth exploration of multi - field synergy. To address this issue, this study closely integrates the characteristics of the internal heat source of the motor, constructs a multi - physical - field coupling model, and quantitatively analyzes the influence law of heat - dissipation fin parameters on heat transfer performance. By realizing the collaborative optimization of the response surface method and the genetic algorithm, a design scheme for the composite heat dissipation structure is proposed, aiming to balance heat dissipation efficiency and material cost and provide innovative ideas for the reliable operation and energy - efficiency improvement of new energy motors [1].

2. Numerical Modeling Study

2.1. Analysis of heat source of new energy motor

The heat source of new energy motor is mainly composed of electromagnetic loss and mechanical loss, and its accurate modeling is the basis of heat dissipation design. Electromagnetic loss includes copper loss and iron loss, of which the copper loss originates from the Joule heat caused by the winding current, which is mathematically expressed in Equation (1):

$$P_{cu} = I^2 R \tag{1}$$

Where I is the rms value of the winding current, R is the winding AC resistance, the value of which is affected by the skin effect and the proximity effect. The iron loss consists of hysteresis loss and eddy current loss of silicon steel sheet under alternating magnetic field, which is described by the modified model of classical Steinmetz equation (2):

$$P_{fe} = K_h f B_m^{\alpha} + K_e (f B_m)^2 \tag{2}$$

Where, K_h and K_e are hysteresis and eddy current loss coefficients, f is the magnetic field frequency, B_m is the flux density amplitude, and α is the material property index. The mechanical loss contains bearing friction and wind resistance loss, which has a nonlinear relationship with the rotational speed, and the expression is Equation (3):

$$P_{mech} = K_f \omega + K_v \omega^3 \tag{3}$$

Where ω is the rotor angular velocity, and Kf and Kv are the friction and wind resistance coefficients, respectively. The above model can quantify the heat source distribution characteristics of the motor under steady state and transient conditions by coupling the electromagnetic field and thermal field simulation, and provide boundary conditions for the subsequent multi-physics field analysis [2].

2.2. CFD model construction

2.2.1. Control equations

The core of computational fluid dynamics (CFD) model construction lies in solving the governing equations describing the behavior of fluid flow and heat transfer. The Navier-Stokes equations are differential equations describing the conservation of momentum in Newtonian fluids, and their vector form is given by Equation (4):

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u u) = -\nabla p + \nabla \cdot \tau + \rho g \tag{4}$$

Where ρ is the fluid density, u is the velocity vector, p is the static pressure, τ is the viscous stress tensor, and g is the volume force. This equation reveals the dynamic equilibrium of inertial forces, pressure gradient, viscous forces and external forces of the fluid through conservation of momentum [3]. The energy conservation equation then describes the heat energy transfer process with the expression of Equation (5):

$$\frac{\partial(\rho h)}{\partial t}+\nabla\cdot(\rho uh)=\nabla\cdot(k\nabla T)+S_h \tag{5}$$

Where h is the specific enthalpy, k is the fluid thermal conductivity, T is the temperature, and Sh is the heat source term. The left side of the equation characterizes the convective term and the right side contains the heat conduction and internal heat source contributions, which is suitable for the coupled solution of the temperature field in the heat dissipation of the motor.

Turbulence simulation requires the introduction of the Reynolds-averaged Navier-Stokes (RANS) equations, where the standard k-ε model closes the set of equations Eqs. (6) and (7) by solving the transport equations for the turbulent kinetic energy, k, and the turbulent dissipation rate, ε. The equations are used as a basis for the simulation of the turbulence:

$$\frac{\partial(\rho k)}{\partial t}+\nabla\cdot(\rho uk)=\nabla\cdot(\frac{\mu_t}{\sigma_k}\nabla k)+G_k-\rho\varepsilon \tag{6}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\nabla\cdot(\rho u\varepsilon)=\nabla\cdot(\frac{\mu_t}{\sigma_\varepsilon}\nabla\varepsilon)+C_{1\varepsilon}C_k\frac{\varepsilon}{k}-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{7}$$

Where μ_t is the turbulent viscosity, G_k is the turbulent kinetic energy generated by the mean velocity gradient, and $C_{1\varepsilon}$, $C_{2\varepsilon}$, σ_k, σ_ε are empirical constants. The model is based on the vortex viscosity assumption and correlates the Reynolds stress with the mean strain rate through the Boussinesq approximation, which is suitable for engineering simulations of complex turbulent flows in the heat dissipation channel of electric motors. The wall function is used to handle the flow in the near-wall region, ensuring a balance between boundary layer resolution and computational efficiency [4].

2.2.2. Meshing and Boundary Conditions

Mesh generation is a crucial step in building a CFD model, directly affecting the solution accuracy and computational efficiency. In view of the geometric complexity of the heat dissipation channels in new - energy motors, a hybrid mesh strategy is adopted: structured meshes are used for regular regions (such as the surfaces of heat dissipation fins), and unstructured meshes are adapted to curved surfaces and slit regions to ensure high - resolution capture of the boundary - layer flow. The meshes near the wall are refined to accurately simulate the changes in the velocity gradient and temperature field. The quality of the global mesh is optimized through indicators such as orthogonality and aspect ratio to avoid numerical divergence caused by distorted elements.

Boundary conditions need to truly reflect the actual working conditions. The inlet boundary is usually set as a velocity inlet or a mass - flow inlet. The flow velocity is determined according to the forced - convection requirements of the cooling medium, and the temperature is given based on the environmental or cooling - system design values. The outlet boundary adopts the pressure - outlet condition, allowing backflow to adapt to complex flow patterns. In the wall conditions, the surface of the heat - dissipation structure is defined as a no - slip boundary. The heat - flux density distribution is mapped based on the heat - source calculation results of the motor loss model to ensure the physical consistency of energy transfer [5]. The temperature boundary conditions are divided into two categories: constant temperature and convective heat dissipation. The latter simulates the natural or forced - cooling effect by combining the surface heat - transfer coefficient.

The heat - flux density boundary condition is directly related to the spatial distribution of the internal losses of the motor. Driven by the coupled electromagnetic - thermal field simulation data, it realizes the collaborative solution of multiple physical fields. The grid independence verification is completed by gradually refining the grid and monitoring the convergence of key parameters (such as pressure drop and temperature extreme values) to ensure that the simulation results are not affected by discretization errors.

2.3. Multi-physical field coupling methods

The multi - physical - field coupling method aims to analyze the dynamic interactions among the fluid, solid, and thermal fields, providing a global simulation framework for the heat dissipation design of new - energy motors. Fluid - structure interaction (FSI) focuses on the bidirectional influence between fluid dynamics and solid deformation. The implementation process is based on the partitioned iterative strategy: the Navier - Stokes equations are used in the fluid domain to solve the distribution of flow - field pressure and shear force, the equations of elasticity are employed in the solid domain to calculate the deformation response, and the data at the interface are transferred through the interpolation algorithm to ensure the continuity of displacement and stress. The weak - coupling strategy solves the fluid and solid equations step - by - step and iteratively updates the boundary conditions until the residuals converge, which is suitable for the analysis of steady - state or quasi - steady - state working conditions [6]. A schematic diagram of the fluid - structure interaction is shown in Figure 1.

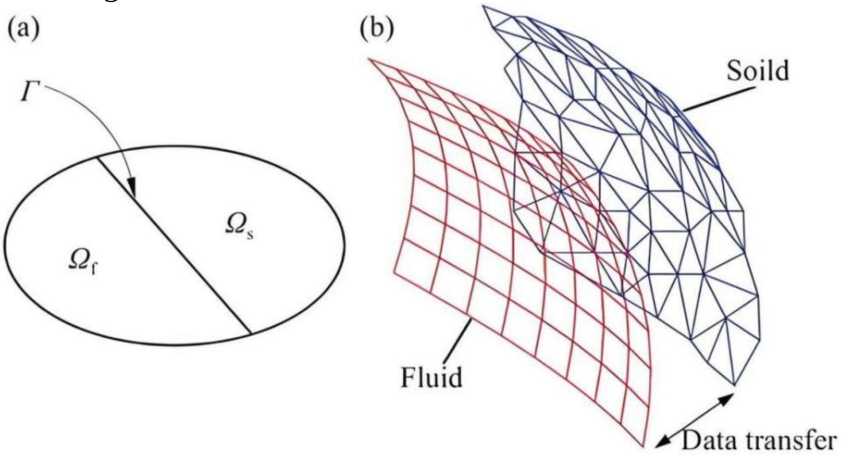

Figure 1 Schematic diagram of flow-solid coupling

Heat-flow coupling focuses on the interaction of fluid heat transfer and temperature field evolution in either one-way or two-way coupling mode. In unidirectional coupling, the fluid flow characteristics drive the heat transfer calculations, ignoring the effect of temperature changes on the flow field, as shown in Figure 2.

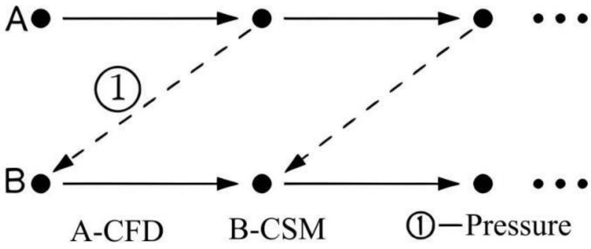

Figure 2 Unidirectional coupling

Bidirectional coupling, on the other hand, solves the energy equation and the flow equation synchronously, which is suitable for the scenarios of natural convection or significant change in viscosity caused by high heat loads, as shown in Fig. 3. In the heat-flow coupling process, the heat source distribution is provided by the electromagnetic loss model, and the flow field parameters are updated by the energy equation to update the temperature field, forming a closed-loop feedback [7].

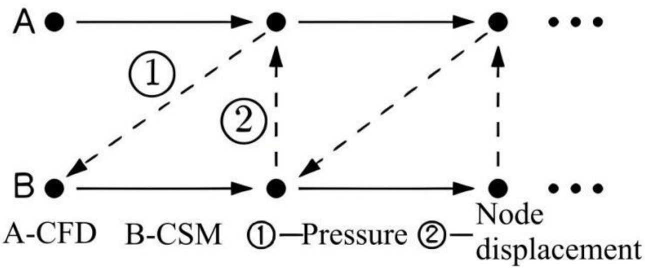

Figure 3 Bidirectional coupling

Multi - field collaborative simulation needs to address the issues of time - scale differences and grid matching. The sequential coupling method is adopted for the fluid - structure - thermal three - field coupling. First, the electromagnetic - thermal field is solved to obtain the heat - source distribution, and then the fluid - structure coupling analysis is driven. The iteration accuracy is controlled through the residual threshold. Commercial software such as ANSYS Multiphysics uses a collaborative simulation engine to achieve cross - module data exchange, ensuring the integrity of the energy - transfer path between physical fields. The grid consistency requires the use of compatible discretization formats at the interface to avoid the accumulation of interpolation errors. Meanwhile, the time - step matching strategy guarantees the stability of transient analysis. This method has verified, through experimental validation and comparison with single - field simulations, the importance of multi - field coupling for the optimization of heat - dissipation structures.

3. Optimized design of heat dissipation structure

3.1. Optimization of fin parameters

The structural parameters of the radiator fins directly affect the equilibrium relationship between air-side flow resistance and heat transfer efficiency, and the optimal solution needs to be sought through a multi-objective optimization method. Among the geometrical characteristics of the staggered fins of a plate-fin radiator, the fin pitch, height and staggered length are the core design variables, the combination of which determines the turbulence intensity in the flow channel, the boundary layer development, and the frequency of the thermal boundary layer re-initiation. Too small a pitch can easily lead to a surge in flow resistance, while too large a pitch can weaken the disturbance effect; fin height affects the trade-off between heat transfer area and flow pressure drop; stagger length regulates the degree of transverse mixing of fluids, which indirectly affects convective heat transfer coefficients [8].

The multi - objective optimization framework based on the genetic algorithm can efficiently search the parameter space and avoid the local - optimum trap. The algorithm takes the heat - transfer performance (such as the Nusselt number) and the flow resistance (such as the pressure - drop coefficient) as the optimization objectives, the fin pitch, height, and staggered - fin length as the decision variables, and sets the overall size of the radiator and the fin thickness as fixed constraints. The fitness function synthesizes the weighted relationship of the objective functions or the Pareto dominance relationship, and iteratively generates the non - dominated solution set through selection, crossover, and mutation operations. When the main program is run in Matlab, the obtained Pareto front solution set is shown in Figure 4.

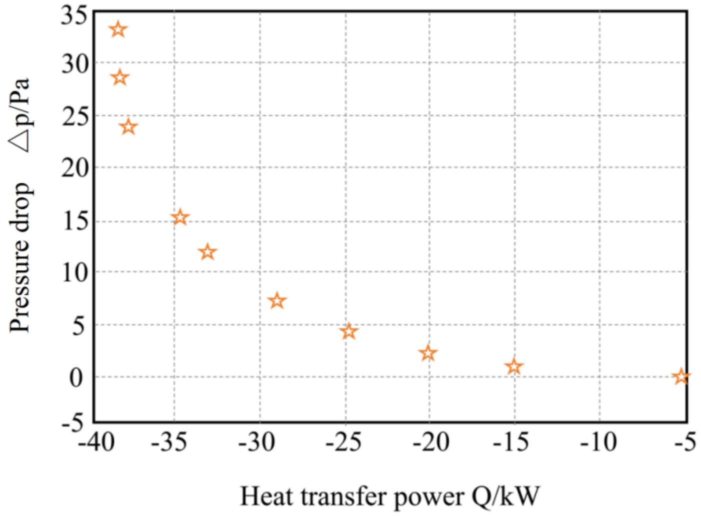

Figure 4 Multi-objective optimization results

The Pareto front solution set characterizes the trade - off relationship where heat transfer and resistance cannot be optimized simultaneously, and a compromise solution needs to be selected according to engineering requirements. Typical selection criteria include the allowable pressure - drop threshold under the maximum heat - transfer efficiency or the highest heat - transfer performance within a specified pressure - drop range. The optimized fin parameter combination can reconstruct the velocity - field and temperature - field distributions in the flow channel. The secondary flow induced by the staggered - tooth structure enhances fluid disturbance, delays the thickening of the thermal boundary layer, and at the same time avoids the energy loss caused by flow separation.

Based on the results of the multi - objective optimization, some fin parameters with good heat transfer and resistance characteristics are selected as shown in Table 1. From the data in the table, it can be seen that the optimized staggered - tooth fin structure can reduce the air - side pressure drop and improve the comprehensive performance of the radiator, providing technical support for the lightweight and efficient heat dissipation design of new - energy motors [9].

Table 1 Radiator construction parameters

Parameter	1	2	3
Heat transfer power Q/kW	5	15	23
Pressure drop △P/Pa	27	35	38
Pitch p /mm	2.5	3.5	3.8
Height h/mm	4	4.5	5
Length l/mm	4	5	6

3.2. Optimization based on the combination of Response Surface Methodology (RSM) and Genetic Algorithm (GA)

The synergistic optimization strategy of response surface methodology and genetic algorithm combines the efficiency of the agent model with the global search capability, which is suitable for the multi-objective design of complex heat dissipation structures. Response surface methodology constructs approximate explicit models of design variables and objective functions through design of experiments (DOE) sampling, which significantly reduces the computational cost. For the optimization of fin geometry parameters, Box-Behnken or central composite design is used to generate the sample space, CFD simulation is used to obtain the heat transfer coefficient and pressure drop response values, and second-order polynomials are used to fit the response surface function, which quantifies the nonlinear effects of pitch, height, and misalignment length on the performance.

The genetic algorithm uses the response surface model as the fitness function to avoid directly invoking highly time-consuming simulations. The initial population randomly generates the fin parameter combinations, the selection operation screens the non-inferior solutions based on the Pareto dominance relation, the crossover and mutation operation explores the parameter space, and the iteration approaches the global optimum. In multi-objective optimization, the competitive relationship between heat transfer efficiency and flow resistance is balanced by weight assignment or constraints to ensure that the solution set meets the actual needs of engineering.

The model accuracy is verified by decision coefficient and residual analysis, and the local sensitivity analysis identifies the key design variables and guides the parameter range correction. The optimization results need to be double-checked by high-fidelity CFD models to ensure that the response surface proxy error is below the threshold. The method shortens the design cycle by decoupling the modeling and optimization processes, while avoiding the local optimization defects of the traditional trial-and-error method. In engineering applications, the fin parameters need to be adjusted with the constraints of the stamping process to ensure the geometric feasibility.

3.3. New composite heat dissipation structure design

The synergistic heat dissipation mechanism of liquid cooling and phase change material (PCM) solves the temperature fluctuation problem in transient high heat flow density scenarios by coupling the advantages of active cooling and passive heat storage. The liquid cooling system is based on the principle of forced convection heat transfer (Fig. 5), where the coolant flows through microchannels or cold plates to enhance the convection heat transfer coefficient through turbulence perturbation to rapidly export the heat from local hot spots; the PCM utilizes the latent heat of the solid-liquid phase change to absorb the peak heat load and slow down the rate of temperature rise. In the composite structure design, PCM is encapsulated between the liquid cooling channels or embedded inside the heat dissipation substrate, which ensures efficient contact between the phase change interface and the liquid cooling medium, and the heat is transferred through the dual paths of conduction and convection.

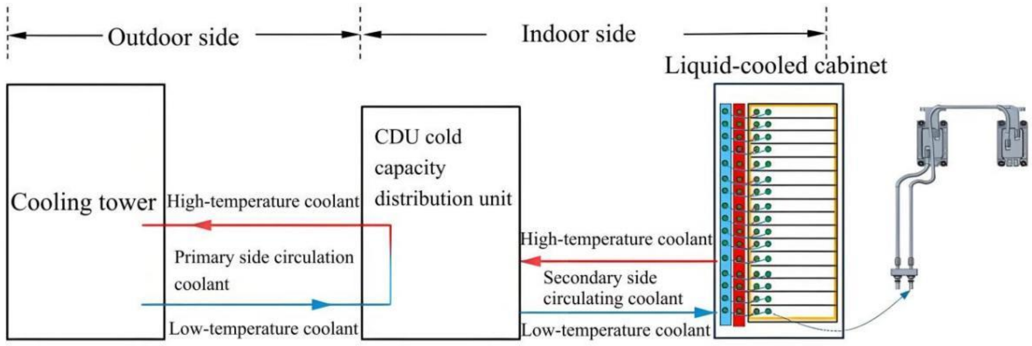

Figure 5 Schematic diagram of a cold plate liquid cooling system

The synergistic heat dissipation mechanism needs to solve the problem of coupled multi-physics field modeling. The flow and heat transfer in the liquid-cooled flow channel are solved by the Navier-Stokes equations in conjunction with the energy equations, and the PCM phase change process is described by the enthalpy-porosity method to describe the apparent specific heat change in the solid-liquid phase region, which avoids the explicit tracking of the phase interface. Conjugate heat transfer boundary conditions are set at the flow-solid interface to ensure the continuity of temperature field and heat flow density. The thermophysical anisotropy of the phase change material is approximated by an equivalent thermal conductivity model, and the heat storage and release rates are dynamically adjusted in combination with the transient temperature distribution of the liquid-cooled work material.

Structural optimization requires balancing thermal storage capacity and flow resistance constraints. The topology optimization method defines the spatial distribution of the PCM-filled region to minimize the peak temperature and pressure drop as the objective function, and the constraints include the volume fraction of PCM and the geometry of the flow channel. The selection of phase change materials is based on the principle of matching the melting temperature and heat load to avoid the phase change hysteresis or overcooling phenomenon affecting the heat dissipation stability. In the experimental validation, the composite structure shows lower temperature rise and more uniform temperature field distribution under the pulsed heat source condition, which confirms the technical advantages of the liquid-cooling-PCM synergistic mechanism in the intermittent high-power heat dissipation scenario [10].

4. Comparative analysis of experiments

The experimental validation is based on the standard “Automobile and Tractor Radiator Blower Test Methods” to build a hydrodynamic test platform, and the test principle is shown in Figure 6. The experimental conditions cover the typical flow rate range, and the test results are compared with the output of the CFD simulation model to verify the prediction ability of the numerical method on the flow and heat transfer characteristics of the finned runner.

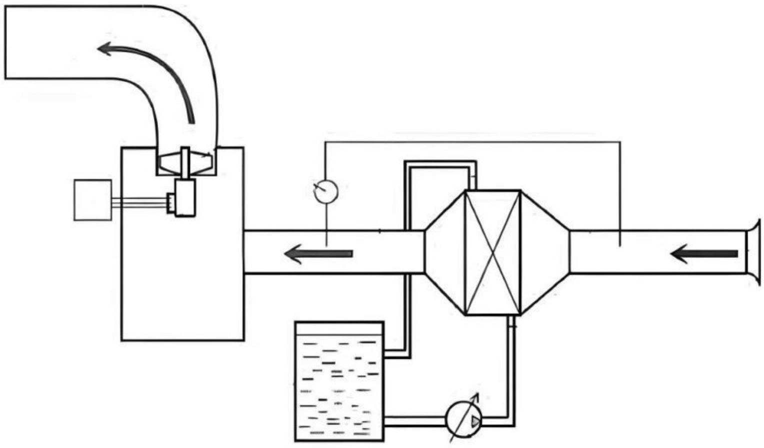

Figure 6 Schematic diagram of the test

The CFD simulation uses the finite volume method to solve the Navier-Stokes equations and the energy equations, the turbulence model selects Realizable k-ε equations to close the Reynolds averaged system of equations, and the wall area uses Scalable Wall Functions to correct the velocity and temperature boundary layer gradients. The finned runner geometry model is parametrically reconstructed based on the actual dimensions, and the boundary layer mesh satisfies the turbulence model applicability condition that the y+ value is less than 5. The conjugate heat transfer model couples the convective heat transfer in the fluid domain with the heat conduction in the solid domain, and the continuity of heat flow at the interface is related to Newton's cooling law through Fourier's law. The simulation results are extracted from the flow channel pressure drop and local

Nussel number distribution, which are verified by quantitative comparison with experimental data.

Error traceability focuses on numerical model assumptions deviating from the experimental system. Mesh-independent validation confirms the convergence of the solution results through a three-level mesh encryption strategy to rule out the possibility of discrete error dominance. Comparison of the pressure drop curves shows that the simulation and experimental trends are in agreement, and the local deviations are attributed to geometrical variations due to fin machining tolerances or sensor nonlinear errors. The temperature field differences are related to the fact that radiative heat dissipation is not modeled, and the effect of radiative heat transfer needs to be corrected by introducing a surface radiation term in the energy equation.

The validation of the genetic algorithm and CFD co-optimization framework relies on the experimental-simulation error controllability. From the comparison curves in Fig. 7, it can be seen that in the velocity interval of 2 m/s~12 m/s, the overall error of the pressure drop curves of the air passing through the fins between the simulated and experimental values is relatively small, and the optimized combinations of fin parameters are introduced into the experimental test after CFD pre-evaluation, which shows that the synchronization between the pressure drop reduction and the improvement of the heat transfer efficiency is in line with the engineering expectation. The error threshold is set as an acceptable range for engineering, and if it is exceeded, the fitting accuracy of the proxy model or the calibration error of the boundary conditions need to be retraced. The method builds a closed-loop link between heat sink performance prediction and design through cross-validation of multi-physical field data, providing reliable method support for the development of heat sink systems for high power density equipment.

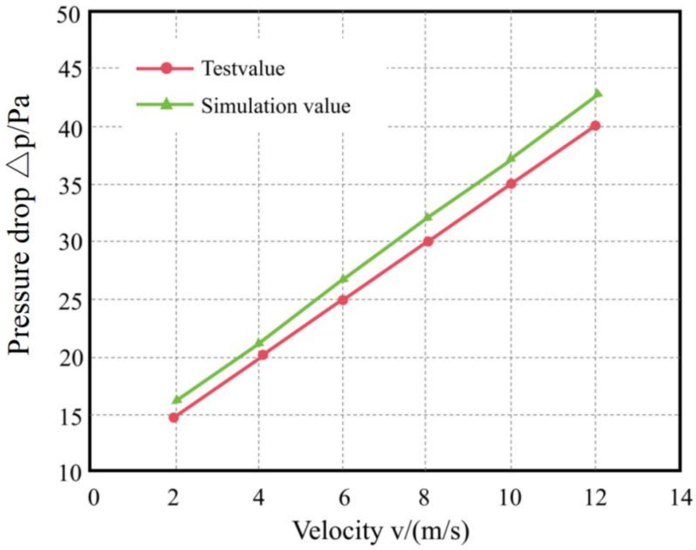

Figure 7 Air-side pressure drop test results

5. Conclusion

It has been confirmed that computational fluid dynamics (CFD) technology has the ability to effectively guide the optimized design of heat dissipation structures for new energy motors. By analyzing the heat flow transfer mechanism through the multi-physics coupled model and combining it with intelligent algorithms to realize the global optimization of parameters, the comprehensive performance of the heat dissipation system has been significantly improved. The new composite heat dissipation structure combines both lightweight and high thermal efficiency, providing a practical solution for engineering practice. In the future, it is necessary to further explore the effects of the synergy between material properties and dynamic working conditions, and expand the application boundary of the multi-objective optimization framework, so as to promote the new energy motor thermal management technology towards the direction of intelligent and sustainable development.

References

[1] Torres R A, Dai H, Jahns T M, et al. Cooling design of integrated motor drives using analytical thermal model, finite element analysis, and computational fluid dynamics[C]//2021 IEEE Applied Power Electronics Conference and Exposition (APEC). IEEE, 2021: 1509-1509.

[2] SanAndres U, Almandoz G, Poza J, et al. Design of cooling systems using computational fluid dynamics and analytical thermal models[J]. IEEE Transactions on Industrial Electronics, 2013, 61(8): 4383-4391.

[3] Sultan U, Ni M, et al. Design optimization with computational fluid dynamic analysis of β-type Stirling engine[J]. Applied Thermal Engineering, 2017, 113: 87-102.

[4] Huang H, Sun Y, Zhang T, et al. Analysis and optimization of oil cooling structure for electric vehicle power motor[J]. Journal of Thermal Science and Engineering Applications, 2025, 17(6): 061007.

[5] Sundén B. Computational fluid dynamics in research and design of heat exchangers[J]. Heat Transfer Engineering, 2007, 28(11): 898-910.

[6] Anderson D, Tannehill J C, Pletcher R H, et al. Computational fluid mechanics and heat transfer[M]. CRC press, 2020:12.

[7] Li W, Yang Q, Yang Y, et al. Optimization of pump transient energy characteristics based on response surface optimization model and computational fluid dynamics[J]. Applied Energy, 2024, 362: 123038.

[8] Norton T, Tiwari B, Sun D W. Computational fluid dynamics in the design and analysis of thermal processes: a review of recent advances[J]. Critical reviews in food science and nutrition, 2013, 53(3): 251-275.

[9] Xu Y, Tan L, Yuan Y, et al. Numerical simulation on flow field and design optimization of a generator unit based on computational fluid dynamics analysis[J]. Mathematical Problems in Engineering, 2021, 2021(1): 3350867.

[10] Fan J, Eves J, Thompson H M, et al. Computational fluid dynamic analysis and design optimization of jet pumps[J]. Computers & Fluids, 2011, 46(1): 212-217.