Research Article

Numerical Simulation on Flow Field and Design Optimization of a Generator Unit Based on Computational Fluid Dynamics Analysis

Yingying Xu,¹ Libin Tan ,^{1,2} Yuejin Yuan ,¹ and Man Zhang¹

¹College of Mechanical and Electrical Engineering, Shaanxi University of Science and Technology, Shaanxi, Xi'an 710021, China

²Loncin Motor Co., Ltd., ChongQing 400039, China

Correspondence should be addressed to Libin Tan; 1364979930@qq.com and Yuejin Yuan; yyjyuan1@163.com

Received 13 April 2021; Revised 5 July 2021; Accepted 14 July 2021; Published 26 July 2021

Academic Editor: Sivasankaran Sivanandam

Copyright © 2021 Yingying Xu et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

Achieving an optimal cooling of the generator unit is indispensable for high working performance. In this work, computational fluid dynamics analysis on the flow field of the conventional type and silent type of the S688CCS series generator unit is conducted, and design optimization of the generator unit with poor cooling is studied based on flow field analysis results. Flow field simulation results indicate that the total cooling air quantity of the silent type generator unit is lower than that of the conventional type generator unit, and the cooling air quantity of the radiator is also lower than that of the conventional type generator unit, which is not conducive for the cooling of the silent type generator unit. The flow field optimization is achieved by adopting the single variable control method to improve the structure of the fan cover, silent components for silence, adjacent structure, and air inlet grille. The corresponding structure optimization scheme is put forward. After optimization, the air quantity for the cooling radiator of the silent type generator unit is 44.33% higher than that of its original structure, and the total cooling air quantity is higher than that of the conventional type generator unit. The research results in this work can provide a theoretical basis for the design of the cooling air flow path of generator units for achieving an optimal cooling performance.

1. Introduction

With the development of science and technology, China's economy and people's living are improved. Due to its low cost and convenience, generator units are widely used in shopping halls, schools, hospitals, and other public places. Therefore, generator units are facing the strategic transformation of rapid demand and expansion. The main components of the general generator unit include engine, magneto, muffler, air filter, starter, radiator, cover parts, framework, and decorative parts, among which the cooling fan and radiator are used for the cooling of the generator unit [1]. The generator unit is usually divided into the conventional type model and silent type model. The most important design challenge of the silent generator unit is how to reasonably design the silencing components of the generator unit under the condition of satisfying aerodynamic acoustics and cooling performance of the generator unit, so as to ensure the feasible operation of the generator

unit. However, better prototype model of the generator unit can be obtained only through several experimental trials, which will delay development time of the new generator unit and waste human and financial resources. Therefore, in the early conceptual design, theoretical analysis and performance evaluation of the design model of the generator unit are of great significance to obtain a reasonable design.

Nowadays, with the rapid development of computer science and technology, the virtual design and development of industrial products based on the integrated analysis platform of virtual simulation have been gradually becoming the trend of the industry [2–5]. For example, Sui [6] has investigated the flow velocity and pressure field distribution of the electrical generator through CFD analysis, providing data support for product performance evaluation. Tan et al. [7, 8] used STAR-CCM+ to conduct the temperature field analysis of the muffler cover used in the generator unit. Meanwhile, in his following research, he conducted a fluid-solid coupling analysis for velocity field and temperature

field analysis of a small rated power silent generator unit. Smaili et al. [9] used the numerical simulation method to investigate cabin thermal performance of wind turbines for giving guidance to heat dissipation optimization. Chen et al. [5] developed an advanced simulation technology by integrating computational fluid dynamics (CFD) and a thermoelectric module (TEM) for analyzing the influence of heat transfer characteristics on generator cooling performance. Singh et al. [10, 11] used CFD simulation to analyze the cooling system of the engine used in the generator and conducting the centrifugal fan design for improving engine cooling performance. Xiao et al. [12] studied the influence of natural wind on the thermal fluid performance of the 300 MW air-cooled cogeneration unit and obtained the optimal operation design of the air-cooled condenser for better cooling performance of the cogeneration unit. Han and Jung [13] analyzed the thermal effect of the electronic unit based on five thermal analysis models and compared the time consumed by different analysis models for accurate thermal analysis. Ma et al. [14] put forward three different heat dissipation layout structures of bottom to side, side to tail, and side to side on the basis of the structure of bottom to tail for obtaining the optimal heat dissipation layout structure of the wind turbine. Borkowski et al. [15] used CFD software Fluent to simulate the mechanical power losses in the hydro-set gap for the hydro-set that consists of the guide vanes and propeller turbine integrated with the permanent magnet synchronous generator. Rüttgers et al. [16] used CFD technology to analyze the flow field over the DrivAer fastback vehicle model for capturing flow separation, vortical structures, and unsteady quantities. Dang et al. [17] investigated the turbulent flow and heat transfer characteristics of the rotor-stator system of a hydro-generator scale model by CFD technology with the conjugate heat transfer model. Through a general review about CFD application in engineering applications, it shows that CFD technology has obvious advantages in quickly and accurately predicting related evaluation performance parameters. Though the CFD results theoretically guide the design and optimization process, the improvement for obtaining good performance of engineering products can be obtained easily [18, 19].

In recent years, many researchers have conducted the flow field characteristics analysis of generator unit or wind turbines used in industrial applications. However, the relative literature about the flow field analysis and cooling performance optimization study on the generator unit used for emergency power supply is rarely. Through CFD modeling and optimization, an optimal cooling design of the generator unit can be quickly obtained. Therefore, the numerical simulation on flow field characteristic and design optimization of generator units is conducted based on computational fluid dynamics (CFD) analysis. The main purpose of this study is to establish a reliable CFD model for obtaining flow field characteristics of the conventional type and silent type of S688CCS series generator unit, verify the effectiveness of the CFD model by air velocity measurement, and conduct design optimization research of the generator unit with poor cooling performance. Through a thorough investigation of fluid flow analysis and cooling

structure optimization, it can form a theoretical reference for the cooling structure design of the generator unit and give the simulation data support for the designers to make a proper evaluation on cooling performance of the generator unit.

2. CFD Modeling of Generator Unit

2.1. Physical Model. Figures 1 and 2 show the three-dimensional models of the conventional type generator unit and silent type generator unit (hereinafter referred to as the conventional type and silent type, respectively). The shape and size for these two types of generator units are basically the same. The length, width, and height of two types of generator units are 5 m, 2 m, and 2.5 m, respectively. The total weight of the generator unit is about 7000 kg, and the rated power of the generator unit is 500 kW. The air cooling duct sections in Figures 1 and 2 are used to monitor the air quantity of each air duct during the simulation calculation and prepare the analysis data for evaluating the cooling effect of generator units. The structure and dimensions of engine, motor, cooling water tank, radiator, and other parts used in the conventional and silent types of the Cummins 6C diesel generator unit are exactly the same. The structural differences between the silent type and the conventional type of the diesel generator unit are air inlet grilles position and the silencing materials of inlet and outlet. In order to achieve a better silent effect, the silent type generator unit has added the corresponding supermute sound-absorbing material, so as to ensure that the mute effect of the silent type generator unit is better than that of the conventional type.

Figure 3 shows the mesh model of two generator units (conventional type and silent type). Mesh generation, solving, and results postprocessing are all completed in CFD software STAR-CCM+ 11.06 (Siemens PLM Software Company, Germany). The polyhedral mesh technology and prim layer mesh technology available in STAR-CCM+ 11.06 are used to generate the mesh model of two generator units and grid refinement for the rotating region and rotating fan tip region which are set by the grid refinement method with smaller mesh size [20, 21]. After mesh generation, the grid number for the computational domain of the generator unit is about 20 million. The grid independence study has proved that the grid number used in this work can give a grid independent solution. Figure 4 shows the plane sections for extracting and counting the air quantity of the radiator and motor.

2.2. Mathematical Model. The fluid flow satisfied three conservation laws: mass conservation law, momentum conservation law, and energy conservation law. When the flow is turbulent, the whole system must follow the turbulent transport equation. The mathematical descriptions of these conservation laws were collectively referred to as control equations. In this paper, the $k-\varepsilon$ turbulence model provided in STAR-CCM+ was used for numerical calculation and the wall function method is set by two-layer all $Y+$ wall

FIGURE 1: Conventional type generator unit.

FIGURE 2: Silent type generator unit.

treatment recommended in STAR-CCM+. It was assumed that the flow in the generator unit is incompressible and steady, and the influence of liquid temperature is not considered in this work.

Therefore, the governing equations for fluid flow (continuity equation and momentum equation) and transport equations for $k-\varepsilon$ turbulence model equations are described in detail below [22–24].

FIGURE 3: Mesh model of two generator units. (a) Conventional type generator unit. (b) Silent type generator unit.

FIGURE 4: Cross section of the radiator and elector for the generator unit.

(1) Continuity equation:

$$\frac{\partial u_i}{\partial x_i} = 0. \quad (1)$$

(2) Momentum equation (N - S equation) [25, 26]:

$$\frac{\partial(u_i u_j)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\left(\mu + \mu_t \right) \frac{\partial u_i}{\partial x_j} \right] - \frac{\partial p}{\partial x_j} + S_i, \quad (2)$$

where ρ is the density, kg/m^3 , u_i is the velocity components, m/s , x_i is spatial coordinates, m , p is the pressure, Pa , μ is the dynamic viscosity, $\text{Pa}\cdot\text{s}$, μ_t is the turbulent viscosity coefficient, $\text{N}\cdot\text{s/m}^2$, and S_i is the source term. In addition, i and $j = 1, 2$, and 3 indicate the components in the Cartesian coordinate system.

(3) Turbulence model [27, 28]:

$$\begin{aligned} \rho \frac{\partial(k)}{\partial t} + \rho u_i \frac{\partial(k)}{\partial x_i} &= \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \\ &+ G_k - \rho \epsilon, \\ \rho \frac{\partial(\epsilon)}{\partial t} + \rho u_i \frac{\partial(\epsilon)}{\partial x_i} &= \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] \\ &+ C_{1\epsilon} \frac{\epsilon}{k} G_k - C_{2\epsilon} \rho \frac{\epsilon^2}{k}, \end{aligned} \quad (3)$$

where k is the turbulent kinetic energy, ϵ is the turbulent dissipation rate, $\mu_t = \rho C_{\mu} k^2 / \epsilon$ is the turbulent viscosity, G_k is the turbulent kinetic energy term generated by the velocity gradient, $C_{1\epsilon}$ and $C_{2\epsilon}$

are empirical constants, and σ_k and σ_ϵ are the Prandtl number of turbulent kinetic energy and dissipation rate, respectively. For the values of empirical constants, $C_{1\epsilon}$ and $C_{2\epsilon}$ are 1.44 and 1.92, respectively, σ_k and σ_ϵ are 1.0 and 1.3, respectively, and C_{μ} is 0.09 [29, 30].

2.3. Boundary Conditions. As for the CFD analysis method for rotating parts in CFD simulation software, numerical methods such as frozen rotor approach, mixing plane, sliding mesh, dynamic mesh, moving reference frame method, and some others are often used. Among them, sliding mesh and dynamic mesh are used for transient calculation. The frozen rotor approach, mixed plane, and moving reference method are used for steady calculation. In STAR-CCM+, the fan rotation in the steady state is accomplished by a moving reference frame (MRF) modeling technique. In the MRF approach, a separate region enclosing the entire rotation region must be defined, and a rotating reference frame was assigned to that region. Therefore, a constant grid flux will be generated in the appropriate conservation equations [31]. Therefore, in this work, the moving reference frame method is used to model rotation of the radiator cooling fan and motor cooling fan. The rotation speed for the radiator cooling fan and motor cooling fan are 1295 r/min and 1500 r/min, respectively. The fluid property is the air at 25°C standard atmospheric pressure, and the boundary condition for the air filter is mass flow inlet with the flow rate 648 g/s. The environmental domain for external flow field analysis is created according to the actual large-scale unit experimental test workshop space (the test workshop room is about 40 m long, 10 m wide, and 10 m high), the inlet boundary of the external flow field

calculation domain is set as the stagnation inlet, the outlet boundary of the external flow field calculation domain is set as the pressure outlet, and the rest is the boundary condition without the sliding wall surface.

The semi-implicit method for the pressure-linked equation (SIMPLE) is applied to handle the pressure-velocity coupling for all numerical simulations [32]. The pressure-based coupled algorithm is used to solve the governing equations. All the equations are discretized and solved using a high-resolution scheme that essentially is the second-order upwind scheme. The enhanced wall treatment method is used to handle the near-wall phenomena of the generator unit. The fully implicit coupling algorithm based on the finite volume method in STAR-CCM+ software is used for calculation. The advection terms are in high-resolution format, and its convergence accuracy is set as 10^{-5} [33]. Numerical simulation is automatically stopped when all the residuals (continuity, energy, X momentum, Y momentum, Z momentum, turbulent kinetic energy, and turbulent dissipation rate) for governing equations were lower than 10^{-5} for assuring the convergence of the simulation. The number of iterations for each simulation is averagely 6000 steps for achieving the convergence criteria goal.

3. Results and Discussion

3.1. Air Quantity Distribution. Figure 5 shows the air quantity distribution characteristics of two generator units (conventional type and silent type). As shown in Figure 5, total air inlet quantity of the silent type generator unit is smaller than that of the conventional type generator unit, and the air quantity of the radiator is also reduced by 14.9%, which is not conducive for the cooling performance of the generator unit. In the silent type generator unit, for ensuring the silence effect, some sound-absorbing materials are added in cooling air inlet ducts, resulting in high flow resistance for air. Meanwhile, the air ducts' size and positions are different from the conventional type generator unit; this maybe another reason for the reduction of total cooling air quantity for the silent type generator unit. Therefore, based on the flow field analysis results, the design optimization of the silent type generator unit should be conducted for improving its cooling performance. The position of sound-absorbing materials and the rearrangement of cooling air ducts maybe two main optimization directions according to CFD analysis results.

Figure 6 shows the experimental validation for generator unit flow fluid analysis. As shown in Figure 6(a), nine evenly distributed measuring points are used to measure the air inlet velocity passing through middle air inlet grilles. In CFD simulation results, the middle air inlet surface is divided into nine subareas, and the average velocity value of each area is calculated for comparison with the experimental value. The experimental validation results ((Figure 6(b)) indicate that the CFD and experimental air inlet velocity illustrate a good agreement, with the average relative error (relative error = absolute value of the difference between simulation and experiment/experimental

FIGURE 5: Air quantity distribution characteristics of the generator unit.

value) approximately 5%, which verifies the prediction accuracy of the CFD model. The reason for the difference between simulated results and experimental results maybe the difference of the data extraction method. The measurement point is a fixed value at a central point, while the simulated data is the average results of the small divided surface.

3.2. Velocity Distribution. Figure 7 shows velocity distribution of the silent type generator unit and conventional type generator unit (same plane section). The silent type generator unit mainly receives cooling air from the middle air inlets at the front end of the generator unit, and the cooling air must pass through the motor and engine area, causing the temperature of air entering the radiator to be higher than that of the conventional type generator unit, reducing the heat exchange performance, which is not conducive to the cooling during the operation of the generator unit. Meanwhile, the silencing material at the inlet changes the flow path of cooling air, increases the flow resistance, and makes the total air quantity of the silent type generator unit much more less than that of the conventional type generator unit.

3.3. Optimization Process of Air Quantity Distribution. Through flow field comparison analysis of two generator units (conventional type and silent type), it is found that the air quantity distribution characteristics of the silent type generator unit are worse than that of the conventional type generator unit. The structure difference of the silent type generator unit mainly lies in the inlet grille positions and the inlet-outlet silencing materials. In order to explore the influence of these structures on flow field characteristics of the silent type generator unit, a single structure of the silent type

FIGURE 6: Model validation analysis. (a) Sketch of the outdoor experiment for air inlet velocity measurement. (b) Comparison curves between the simulated value and experimental value of air inlet velocity (unit: m/s).

FIGURE 7: Velocity distributions of the generator unit. (a) Silent type generator unit. (b) Conventional type generator unit.

generator unit is modified to judge its influence weight on flow field characteristics, which provide a reference for the subsequent optimization design.

3.3.1. Influence of Inlet and Outlet Silencing Materials. Figure 8 shows the comparison between the air quantity distribution of the silent type generator unit and that of the initial silent type generator unit after removing the inlet silencing material. After removing the inlet silencing materials, the total air quantity of the generator unit increases, the air quantity at the cooling duct of the motor and radiator increases, and the air quantity at the radiator increases by about 11.46%, indicating that the inlet silencing material has a great impact on flow field characteristics of the generator unit. The air flow resistance is reduced after removing inlet silencing materials, resulting in the airflow that is much more smooth and without any barrier. So, the total air quantity of the optimized silent type generator unit is increased obviously.

Figure 9 shows the comparison between the air quantity distribution of the silent type generator unit and that of the initial silent type generator unit after removing the outlet silencing material. Air quantity distribution of the generator unit does not change much after removing the outlet silencing material, and the air quantity at the radiator increases by 0.75%, indicating that the outlet silencing material has no obvious effect on the flow field characteristics of the generator unit. The outlet silencing material is placed at downstream of airflow. The airflow enters into the generator unit from the inlet grille, then passes through the radiator and engine, and finally discharges into atmosphere from outlet. The placement of the outlet silencing material may affect the air outflow resistance but no big influence on its upstream resistance. Therefore, the total air quantity of the silent type generator unit has no obvious change after removing the outlet silencing material.

Figure 10 shows the comparison between the air quantity distribution of the silent type generator unit and that of the initial silent type generator unit after only removing the lower

FIGURE 8: Influence of the inlet silent material on air quantity distribution.

FIGURE 9: Influence of the outlet silent material on air quantity distribution.

baffle of the inlet silencing material. Flow field distribution of the generator unit after modification is different from the initial flow field, and air quantity of the radiator increases by 2.82%. This structure modification can be used as an idea for the subsequent optimization. If the upper baffle of the inlet silencing material is removed in the same way of structural modification, the total air intake of the silent type generator unit is 3.013 kg/s and cooling air quantity at the radiator section is 2.404 kg/s, which is lower than that of the original structure.

3.3.2. Influence of Air Inlet Grille. Figure 11 shows the effect of the location of the middle air inlet grille on air quantity distribution. Changing the air inlet grille to the downward and horizontal mode can improve the total air inlet quantity of the generator unit and cooling air quantity passing through the radiator. The airflow flow resistance at the downward or horizontal grille is smaller than grille face up. The improvement effect of air quantity of the horizontal air inlet grille is slightly more obvious than that of the downward air inlet grille for the reason that the airflow flow resistance of the horizontal air inlet grille is smallest among those three different grilles (up, down, and horizontal).

Figure 12 shows the comparison between the air quantity distribution of the silent type generator unit and that of the initial silent type generator unit after removing the left and right air inlet grilles (that is, the air inlets on the left and right sides are square). The flow field of the adjusted silent type generator unit has no obvious change. In the numerical value, the total air quantity increases by 0.06 kg/s, the total air quantity increase ratio is about 2%, the air quantity at the radiator increases by 0.036 kg/s, and the increase ratio is only about 1.4%.

Figure 13 shows the air quantity comparison diagram after adding the left air inlet grille and right inlet grille to the case shell of the silent type generator unit. It shows that total air quantity of the silent type generator unit increases after left and right air inlet grilles are added, and the cooling air quantity of the radiator also increases, and the increment ratio is approximately 8.4% compared with cooling air passing through the radiator of its initial silent type generator unit, indicating that the addition of left and right air inlet grilles for absorbing fresh air from the external environment has a significant effect on cooling air quantity of the radiator. The total air inlet-grilles area is increased by adding two new air inlet grilles, and it inevitably leads to the increase of the total air quantity of the silent type generator unit.

3.3.3. Influence of Fan Cover. Figure 14 shows the comparison between the air quantity distribution of the silent type generator unit after removing the fan cover in front of the radiator and that of the initial structure. After removing the fan cover in front of the radiator, the total air quantity of the generator unit has more than doubled, and the cooling air quantity passing through radiator has also greatly increased, with 131.47% increment, indicating that the fan cover has a great impact on flow field characteristics of the silent type generator unit. Without fan cover, the airflow flow resistance is almost zero, which causes the airflow become extremely smooth, and the cooling air flow velocity is strengthened, and the total air quantity of the silent type generator unit increases dramatically. Obvious increment of total air inlet quantity is conducive to improving the cooling performance of the generator unit, and the design without the fan cover or reducing the flow resistance induced by the fan cover can be used as a guidance for later optimization.

3.4. Optimization Design. Based on flow field analysis results and structural influential factor investigation, the air quantity distribution of the silent type generator unit is

FIGURE 10: Influence of downward baffle on air quantity distribution. (a) Schematic diagram of removing the lower baffle. (b) Air quantity distribution.

FIGURE 11: Influence of the middle air-inlet grille on wind distribution. (a) Schematic diagram of middle air inlet grille modification. (b) Air quantity distribution.

greatly influenced by the structure of the air inlet silencing material, the horizontal placement of the air inlet grille, the increase of the air inlet grilles, and the structure of the fan

cover. In order to ensure the mute effect of the silent type generator unit, the optimization design only changes the lower baffle of the inlet silencing material. According to CFD

FIGURE 12: Influence of the left-right air-inlet grille on wind distribution.

(a)

(b)

FIGURE 13: Influence of added air-inlet grilles on air quantity distribution. (a) Schematic diagram of adding the left and right air inlet grilles. (b) Air quantity distribution.

analysis results, the following three optimization designs are proposed:

- (1) Optimization design I (opt1): horizontal arrangement of the middle air inlet grille, and removing some baffles of the silencing material at the lower part of the inlet

- (2) Optimization design II (opt2): the left and right air inlet grilles are added on the case shell of the generator unit on the basis of optimization design I
- (3) Optimization design III (opt3): opening more grille holes on the original fan cover, as shown in Figure 15

FIGURE 14: Influence of the fan cover on wind distribution.

FIGURE 15: The sketch for the modification of the fan cover. (a) Origin. (b) Optimized.

Table 1 shows the air quantity of three optimization designs of the silent type generator unit. Three optimization designs can improve the cooling air quantity passing through the radiator. In the optimization design I, the air quantity of the radiator is 7.31% higher than that of the original state, but the air quantity of the radiator is still lower than that of the conventional type generator unit. In the optimization design II, air quantity of the radiator is 12.29% higher than that of the original state. The air quantity of the optimized silent generator unit is basically the same as that of the conventional type generator unit. In optimization III, the air quantity of the radiator is increased by 44.33%, which greatly improves the cooling of the silent type generator unit. In three optimization designs, the air quantity passing through the motor surface is basically the same, indicating

that the optimization designs do not change the flow field of the original motor while increasing the cooling air quantity of the radiator. The silencing material of the silent type generator unit is a very important component, and its modification may affect the noise of the generator unit. All the analysis and optimization in this study are only based on the flow field characteristics of the generator unit, without considering the noise factor. In the further research, the flow field analysis and optimization can be coupled with noise prediction for obtaining a more effective design of the generator unit.

Figure 16 shows the velocity distribution of the fan cover surface. Velocity distribution of the optimized fan cover surface is much more uniform, and more cooling air flows through the opening holes to the radiator. Figure 17 shows

TABLE 1: Air quantity evaluation for the optimized silent type generator unit.

Designs	Total mass of air (kg/s)	Middle air inlet (kg/s)	Left air inlet (kg/s)	Right air inlet (kg/s)	Motor area (kg/s)	Radiator (kg/s)	Air quantity improvement of radiator (%)
Origin	3.202	3.017	1.792	0.614	0.611	0.598	7.31
Opt1	3.202	2.261	0.459	0.482	0.602	2.585	7.31
Opt2	3.279	1.644	0.776	0.859	0.603	2.705	12.29
Opt3	4.677	2.444	0.806	0.822	0.605	3.477	44.33

FIGURE 16: The cell relative velocity distribution of the fan cover. (a) Origin. (b) Optimized.

FIGURE 17: Comparison of air quantity distribution for the generator unit.

the air quantity comparison of generator units. As shown in Figure 17, the total air quantity of the optimized silent type generator unit (optimization design III) is higher than that of the conventional type generator unit, and the cooling air

quantity of the radiator is also higher than that of the conventional type generator unit, indicating that the cooling performance of the optimized silent type generator unit is better than that of the conventional type generator unit.

4. Conclusions

For achieving an optimal cooling of the generator unit, the flow filed analysis of two generator units (conventional type and silent type) is conducted and the optimization design for the silent type generator unit is studied based on the flow field results and structural influential factor analysis. The main conclusions drawn are given as follows:

- (1) The total air quantity of the silent type generator unit is lower than that of the conventional type generator unit, and air quantity of the radiator is also lower than that of the conventional type generator unit. CFD and experimental air inlet velocity illustrate a good agreement with 5% discrepancy. Experimental validation proved that CFD simulation can effectively predict flow field characteristics of the generator unit and provide effective simulation data for cooling performance evaluation.
- (2) Structural influential factor analysis results show that air inlet silencing materials have a great influence on the air quantity distribution. The horizontal arrangement of the middle air inlet grille and the addition of air inlet grilles can improve total air

quantity of the silent type generator unit and air quantity of the radiator. The fan cover in front of the radiator has obvious impact on air quantity distribution of the silent type generator unit.

- (3) The optimal design of adding grille holes on the fan cover has significant effect on cooling performance improvement of the generator unit. The total cooling air quantity of the optimized silent type generator unit is higher than that of the conventional type generator unit, and the cooling air quantity for the radiator is also higher than that of the conventional type generator unit. The cooling air quantity of the radiator is 44.33% higher than that of the radiator under its initial structure, and the total amount of cooling air is increased, which is conducive to the cooling of the silent type generator unit. The research results of this work can provide a theoretical basis for the design of the cooling air duct of the generator unit.

Data Availability

No data were used to support the findings of the study.

Conflicts of Interest

The authors declare that they have no conflicts of interest.

Acknowledgments

The authors thank the Loncin Motor Co. Ltd and its technical staff of CFD team for their support on the development of this research by allowing the use of its facilities (high-performance computing server) and the required technical guidance. The authors thank the support provided by National Natural Science Foundation of China (Grant no. 51876109), Key Project of International Science and Technology Cooperation Program for Shaanxi Province (Grant no. 2020KWZ-015), International Science and Technology Cooperation Program for Shaanxi Province (Grant no. 2018KW-018), Key Research and Development Program of Shaanxi Province (Grant no. 2017KW-001), National Key Research and Development Program of China (Grant no. 2017YFD0400902-1), and The Youth Innovation Team of Shaanxi Universities (2019).

References

- [1] S. Morohoshi and R. Tsuru, *Engine Generator Unit: US, US6331740*, Honda Motor Co Ltd, Tokyo, Japan, 2001.
- [2] T. Seyedsaeed, N. N. B. N. Ghazali, W. T. Chong et al., "Computational and experimental optimization of the exhaust air energy recovery wind turbine generator," *Energy Conversion and Management*, vol. 126, pp. 862–874, 2016.
- [3] D. Sáinz, P. M. Diéguez, J. C. Urroz et al., "Conversion of a gasoline engine-generator set to a bi-fuel (hydrogen/gasoline) electronic fuel-injected power unit," *International Journal of Hydrogen Energy*, vol. 36, no. 21, pp. 13781–13792, 2011.
- [4] Y. J. Dai, H. M. Hu, T. S. Ge, R. Z. Wang, and P. Kjellsen, "Investigation on a mini-CPC hybrid solar thermoelectric generator unit," *Renewable Energy*, vol. 92, pp. 83–94, 2016.
- [5] W.-H. Chen, Y.-X. Lin, Y.-B. Chiou, Y.-L. Lin, and X.-D. Wang, "A computational fluid dynamics (CFD) approach of thermoelectric generator (TEG) for power generation," *Applied Thermal Engineering*, vol. 173, no. 6, Article ID 115203, 2020.
- [6] Y. J. Sui, "Analysis and research of the field flow distribution inside gen-set canopy based on CFD method," *Movable Power Station and Vehicle*, vol. 2013, no. 4, pp. 13–19, 2013.
- [7] L. B. Tan, Y. J. Yuan, C. Huang, and Q. Yu, "Numerical simulation of muffler cover temperature analysis for general purpose engine based on CFD," *Energy Sources Part A Recovery Utilization and Environmental Effects*, 2020.
- [8] L. Tan, Y. Yuan, C. Huang, and Q. Yu, "CFD simulation and investigation on flow field characteristics and temperature predictions of an electric generator unit," *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, vol. 43, no. 5, p. 263, 2021.
- [9] A. Smaili, A. Tah, and C. Massfon, "Thermal analysis of wind turbine nacelle operating in algerian saharan climate," *Energy Procedia*, vol. 18, no. 143, pp. 187–196, 2012.
- [10] O. P. Singh, M. Garg, V. Kumar, and V. Chaudhary, "Effect of cooling system design on engine oil temperature," *Journal of Applied Fluid Mechanics*, vol. 6, no. 1, pp. 61–71, 2013.
- [11] O. P. Singh, T. Sreenivasulu, and M. Kannan, "Influence of geometric parameters on centrifugal fan performance and its significance in automotive applications," *International Journal of Modeling Simulation & Scientific Computing*, vol. 3, no. 3, Article ID 1250007, 2012.
- [12] L. Xiao, Z. Ge, X. Du, L. Yang, and Z. Xu, "Operation of air-cooling CHP generating unit under the effect of natural wind," *Applied Thermal Engineering*, vol. 107, pp. 827–836, 2016.
- [13] C. K. Han and H. Jung, "A study on thermal behaviour prediction for automotive electronic unit based on CFD," in *Proceedings of the 2017 23rd International Workshop on Thermal Investigations of ICs and Systems (THERMINIC)*, Amsterdam, Netherlands, September 2017.
- [14] W. Q. Ma, D. B. Sun, Y. Y. Su, and S. R. Wang, "Research on optimal method of heat dissipation layouts of nacelle of wind turbine generator system," *Renewable Energy Resources*, vol. 35, no. 5, pp. 721–726, 2017.
- [15] D. Borkowski, M. Węgiel, P. Ochoń, and T. Węgiel, "CFD model and experimental verification of water turbine integrated with electrical generator," *Energy*, vol. 185, no. 10, pp. 875–883, 2019.
- [16] M. Rüttgers, J. Park, and D. You, "Large-eddy simulation of turbulent flow over the DrivAer fastback vehicle model," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 186, pp. 123–138, 2019.
- [17] D.-D. Dang, X.-T. Pham, P. Labbe, F. Torriano, J.-F. Morissette, and C. Hudon, "CFD analysis of turbulent convective heat transfer in a hydro-generator rotor-stator system," *Applied Thermal Engineering*, vol. 130, pp. 17–28, 2018.
- [18] T. Watanabe, Y. Anoda, and M. Takano, "System-CFD coupled simulations of flow instability in steam generator U tubes," *Annals of Nuclear Energy*, vol. 70, pp. 141–146, 2014.
- [19] V. Hovi, T. Pättikangas, and V. Riikonen, "Coupled one-dimensional and CFD models for the simulation of steam generators," *Nuclear Engineering and Design*, vol. 310, no. 12, pp. 93–111, 2016.
- [20] A. Rynell, M. Chevalier, M. Åbom, and G. Efraimsson, "A numerical study of noise characteristics originating from a

- shrouded subsonic automotive fan,” *Applied Acoustics*, vol. 140, pp. 110–121, 2018.
- [21] S. Wang, T. Avadiar, M. C. Thompson, and D. Burton, “Effect of moving ground on the aerodynamics of a generic automotive model: the DrivAer-estate,” *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 195, Article ID 104000, 2019.
 - [22] A. Pandhare, A. Lal, P. Vanarse, N. Jadhav, and K. Yemul, “CFD analysis of flow through muffler to select optimum muffler model for ci engine,” *International Journal of Latest Trends in Engineering and Technology (IJLTET)*, vol. 4, no. 1, pp. 12–19, 2014.
 - [23] Y. Yuan, L. Tan, Y. Xu, and Y. Yuan, “Three-dimensional CFD modeling and simulation on the performance of steam ejector heat pump for dryer section of the paper machine,” *Vacuum*, vol. 122, pp. 168–178, 2015.
 - [24] P. R. S. Silva, A. J. K. Leiroz, and M. E. C. Cruz, “Evaluation of the efficiency of a heat recovery steam generator via computational simulations of off-design operation,” *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, vol. 42, no. 11, p. 569, 2020.
 - [25] L. Zhang, S. F. Liang, and C. X. Hu, “Flow and noise characteristics of centrifugal fan under different stall conditions,” *Mathematical Problems in Engineering*, vol. 2014, Article ID 403541, 9 pages, 2014.
 - [26] X. Y. Wu, H. Y. Zhang, Z. H. Zhu, and L. M. Song, “Numerical investigation of a liquid cooled battery module with collaborative heat dissipation in both axial and radial directions,” *Journal of Engineering Thermophysics*, vol. 41, no. 7, pp. 1784–1791, 2020.
 - [27] Z. J. Wang, J. S. Zheng, L. L. Li, and S. Luo, “Research on three-dimensional unsteady turbulent flow in multistage centrifugal pump and performance prediction based on CFD,” *Mathematical Problems in Engineering*, vol. 2013, Article ID 589161, 7 pages, 2013.
 - [28] C. V. Silva, L. F. Dondoni, C. Ermel, P. L. Bellani, D. A. Stradioto, and M. L. S. Indrusiak, “CFD analysis of the combustion process in a boiler of a 160 MWe power plant: leakage influence,” *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, vol. 41, no. 7, p. 287, 2019.
 - [29] Y. J. Liu, J. Ma, N. Ma, and Z. J. Huang, “Experimental and numerical study on hydrodynamic performance of an underwater glider,” *Mathematical Problems in Engineering*, vol. 2018, Article ID 8474389, 13 pages, 2018.
 - [30] H. Najafi Khaboshan and H. R. Nazif, “Investigation of heat transfer and pressure drop of turbulent flow in tubes with successive alternating wall deformation under constant wall temperature boundary conditions,” *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, vol. 40, no. 2, p. 42, 2018.
 - [31] C. Zhang, M. Uddin, A. C. Robinson, and L. Foster, “Full vehicle CFD investigations on the influence of front-end configuration on radiator performance and cooling drag,” *Applied Thermal Engineering*, vol. 130, pp. 1328–1340, 2018.
 - [32] A. Eltayesh, F. Castellani, M. Burlando et al., “Experimental and numerical investigation of the effect of blade number on the aerodynamic performance of a small-scale horizontal axis wind turbine,” *Alexandria Engineering Journal*, vol. 60, no. 4, pp. 3931–3944, 2021.
 - [33] W. Lei, X. Zhao, W. Li, W. Shi, L. Ji, and L. Zhou, “Numerical prediction and performance experiment in an engine cooling water pump with different blade outlet widths,” *Mathematical Problems in Engineering*, vol. 2017, Article ID 8945712, 11 pages, 2017.